CR 93.003

June 1993

NCEL

Contract Report

An Investigation Conducted by
Adaptive Research Corporation
Huntsville, AL

AD-A267 833

STRUCTURED FINITE VOLUME MODELING OF U.S. NAVY AIRCRAFT ENGINE TEST CELLS

TASK 2: TURBOPROP ENGINE -

FINAL REPORT - VOLUME 1

Abstract This report presents results of the numerical simulation of a U.S. Naval turboprop test cell facility. The ultimate purpose of this simulation was to provide the Navy with a numercial model to be used for the evaluation of the aerothermal performance of test cells. This simulation was performed using the structured finite volume (SFV) computer code. A description of the physical model, mathematical details, boundary conditions, and results of the study are presented and covered in Volume 1.

Volume 2, Code Documentation and Listings, provides a copy of the input files developed for the modeling of turboprop test cells.

93-18644

NAVAL CIVIL ENGINEERING LABORATORY PORT HUENEME CALIFORNIA 93043-4328

METRIC CONVERSION FACTORS

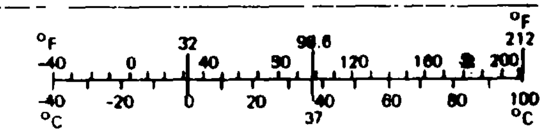
Approximate Conversions to Metric Measures

Symbol	When You Know	Multiply by	To Find	Symbol
		LENGTH		
in	inches	*2.5	centimeters	cm
ft	feet	30	centimeters	cm
yd	yards	0.9	meters	m
mi	miles	1.6	kilometers	km
		AREA		
in^2	square inches	6.5	square centimeters	cm^2
ft^2	square feet	0.09	square meters	m^2
yd^2	square yards	0.8	square meters	m^2
mi^2	square miles	2.6	square kilometers	km^2
	acres	0.4	hectares	ha
		MASS (weight)		
oz	ounces	28	grams	g
lb	pounds	0.45	kilograms	kg
	short tons (2,000 lb)	0.9	tonnes	t
		VOLUME		
tsp	teaspoons	5	milliliters	ml
Tbsp	tablespoons	15	milliliters	ml
fl oz	fluid ounces	30	milliliters	ml
c	cups	0.24	liters	l
pt	pints	0.47	liters	l
qt	quarts	0.95	liters	l
gal	gallons	3.8	liters	l
ft^3	cubic feet	0.03	cubic meters	m^3
yd^3	cubic yards	0.76	cubic meters	m^3
		TEMPERATURE (exact)		
°F	Fahrenheit temperature	5/9 (after subtracting 32)	Celsius temperature	°C

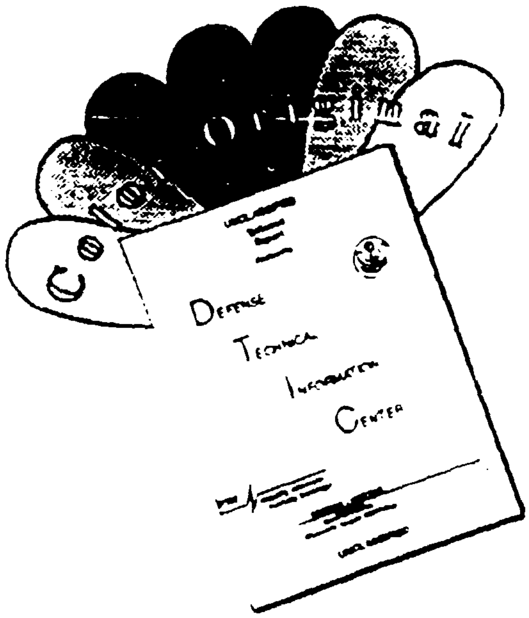
*1 in = 2.54 (exactly). For other exact conversions and more detailed tables, see NBS Misc. Publ. 286, Units of Weights and Measures, Price $2.25, SD Catalog No. C13.10:286.

Approximate Conversions from Metric Measures

Symbol	When You Know	Multiply by	To Find	Symbol
		LENGTH		
mm	millimeters	0.04	inches	in
cm	centimeters	0.4	inches	in
m	meters	3.3	feet	ft
m	meters	1.1	yards	yd
km	kilometers	0.6	miles	mi
		AREA		
cm^2	square centimeters	0.16	square inches	in^2
m^2	square meters	1.2	square yards	yd^2
km^2	square kilometers	0.4	square miles	mi^2
ha	hectares (10,000 m^2)	2.5	acres	
		MASS (weight)		
g	grams	0.035	ounces	oz
kg	kilograms	2.2	pounds	lb
t	tonnes (1,000 kg)	1.1	short tons	
		VOLUME		
ml	milliliters	0.03	fluid ounces	fl oz
l	liters	2.1	pints	pt
l	liters	1.06	quarts	qt
l	liters	0.26	gallons	gal
m^3	cubic meters	35	cubic feet	ft^3
m^3	cubic meters	1.3	cubic yards	yd^3
		TEMPERATURE (exact)		
°C	Celsius temperature	9/5 (then add 32)	Fahrenheit temperature	°F



DISCLAIMER NOTICE



THIS DOCUMENT IS BEST QUALITY AVAILABLE. THE COPY FURNISHED TO DTIC CONTAINED A SIGNIFICANT NUMBER OF COLOR PAGES WHICH DO NOT REPRODUCE LEGIBLY ON BLACK AND WHITE MICROFICHE.

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-018

1. AGENCY USE ONLY *(Leave blank)*

2. REPORT DATE
June 1993

3. REPORT TYPE AND DATES COVERED
Final; December 1990 - September 1992

4. TITLE AND SUBTITLE
STRUCTURED FINITE VOLUME MODELING OF U.S. NAVY AIRCRAFT ENGINE TEST CELLS TASK 2: TURBOPROP ENGINE - FINAL REPORT - VOLUME 1

5. FUNDING NUMBERS
PR - 0604215N
C - N47408-91-C-1228
DN - 661008

6. AUTHOR(S)
P. L. Daley and W. A. Mahaffey

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
Adaptive Research Corporation
4960 Corporate Drive, Suite 100-A
Huntsville, AL 35805

8. PERFORMING ORGANIZATION REPORT NUMBER
CR 93.003

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
Commander / Naval Civil Engineering Laboratory
Naval Air Systems Command / 560 Laboratory Drive
Code 09Y / Facilities Systems Division/L53
Washington, DC 20362-5101 / Port Hueneme, CA 93043-4328

10. SPONSORING/MONITORING AGENCY REPORT NUMBER

11. SUPPLEMENTARY NOTES

12a. DISTRIBUTION/AVAILABILITY STATEMENT
Approved for public release; distribution is unlimited.

12b. DISTRIBUTION CODE

13. ABSTRACT *(Maximum 200 words)*

This report presents results of the numerical simulation of a U.S. Naval turboprop test cell facility. The ultimate purpose of this simulation was to provide the Navy with a numercial model to be used for the evaluation of the aerothermal performance of test cells. This simulation was performed using the structured finite volume (SFV) computer code. A description of the physical model, mathematical details, boundary conditions, and results of the study are presented and covered in Volume 1.

Volume 2, Code Documentation and Listings, provides a copy of the input files developed for the modeling of turboprop test cells.

14. SUBJECT TERMS
Computational fluid dynamics, test cells, aviation test facilities

15. NUMBER OF PAGES
59

16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
Unclassified	Unclassified	Unclassified	UL

TABLE OF CONTENTS

LIST OF FIGURES

1. INTRODUCTION

1.1 Purpose of the Report

This report describes the development and applications of a computer model that simulates steady flow inside a turboprop engine test facility. This model was developed using the Structured Finite Volume (SFV) code [1,2], a general purpose computational fluid dynamics (CFD) code. The report provides details of the mathematical model and reviews the results obtained from the computations. This work was undertaken by Adaptive Research Corporation, for the Naval Facilities Engineering Command, under Contract Number N47408–91–C–1228.

1.2 Background

U. S. Navy aircraft engines completing maintenance or repair are operated in test cells where they must meet performance specifications before reinstallation. Test cells are fully enclosed, sound absorbing hangers. The principle function of these facilities is to provide repeatable, specific test conditions. In the process, the test cells must attenuate the jet and propeller noise sufficiently for personal to carry out normal activities near the facility, cool and control the engine exhaust flow to the point that cell components are not damaged, and meet environmental standards.

The Navy has recently completed construction of the prototypes of a new generation of test cells designed for turboprop and turboprop engines. These test cells, located at the Marine Corps Air Facility, Camp Pendleton, California, and the Naval Air Station, Sigonella, Italy are scheduled to begin operating shortly.

1.3 Objective of the Study

The objective of this work is to provide a computational fluid dynamics model, employing the SFV code, of these prototype engine test cells for use in performance evaluation and for troubleshooting.

1.4 Outline of the Report

This report consists of six main sections. Following this introduction, Section 2 presents the geometric details of the turboprop test cell along with operating conditions. Next, Section 3 contains information about the mathematical basis of the numerical model. In Section 4, the numerical details of the model such as boundary conditions are presented. The results of the present study are presented in Section 5, and the conclusions drawn are given in the final section.

2. DESCRIPTION OF THE SIMULATED PROCESS

2.1 Geometry

Details of the geometry are provided in Appendix A. The engine with propeller is placed in a retangular test bay. Air is drawn into the test bay through the plenum. This air, for the most part, then enters an orifice before contact with the propeller. The two rows of intake vanes, plenum, and orifice provide a smooth, low velocity, low angle of approach airflow to the propeller. Exhaust from the engine, along with some cooler test bay air entrained by the exhaust jet, leave the facility through the augmenter tube, discharging to a chimney and then to the atmosphere. The rest of the through flow is discharged through a set of baffles on the back side of the test bay. A set of acoustic baffles are also located at the top of the chimney.

The engine considered in this modeling effort is the T56 turboprop engine manufactured by Allison Corporation. The operating conditions for the engine are listed in Table 1. The ambient conditions outside the cell were 29.92 mm Hg (1.01325×10^5 N/m^2), 77 F (25 C), and a dry bicomponent atomsphere (76.83 wt per cent N_2).

TABLE 1. T56 Turboprop Engine Operating Conditions

Exhaust Flow	32.4 lb/s 14.7 kg/s
Fuel Flow	0.833 lb/s 0.375 kg/s
Average Exit Temperature	1100 F 600 C
Exhaust Composition (wt%)	0.7479 N_2 0.1411 O_2 0.8100 CO_2 0.3000 H_2O
Prop Speed	1021 RPM
Power Output	4591 SHP

3. MATHEMATICAL FORMULATION

3.1 The Structured Finite Volume (SFV) Mathematical Formulation

The SFV computer code is a general–purpose CFD code in wide use throughout government, industry, and academia. It can model two– or three–dimensional, laminar or turbulent, single– or two–phase fluid flows in arbitrary geometric flow domains. SFV solves the conservation equations for mass, momentum, and energy. The details of this solution procedure follow.

3.1.1 The Governing Equations

The flow field in a given geometry can be described by the conservation equations for mass, momentum, and energy. These equations can be expressed in the following form

$$\frac{\partial}{\partial T} + \nabla (\rho \, \nabla \,) = 0 \qquad (1)$$

$$\frac{\partial(\rho \nabla)}{\partial t} + \nabla \, (\rho \nabla \nabla) = - \nabla p + \nabla \, (\mu_{eff} \nabla \nabla) \qquad (2)$$

$$\frac{\partial(\rho \nabla)}{\partial t} + \nabla \, (\rho \nabla h) = \overline{\nabla q} + \mu_{eff} \nu + \frac{p}{\partial t} + \nabla \nabla P \qquad (3)$$

where

∇	–	is the time–mean velocity;
ρ	–	is the gas density;
p	–	is the static pressure;
μ_{eff}	–	is the "effective" viscosity;
h	–	is the enthalpy;
$\bar{q}$	–	is the diffusive energy flux; and
ϕ	–	is the dissipation function.

The effective viscosity is defined by the relation

$$\mu_{eff}=\mu+\mu_t \;;\; \mu_t=C_\mu \rho \frac{k^2}{\epsilon}$$

where μ is the molecular viscosity and μ_t, the turbulent viscosity, is deduced by employing a turbulence model. SFV employs the two-equation k–ϵ turbulence model [3]. This treatment of turbulence requires the solution of two additional partial differential equations

$$\frac{\partial(\rho k)}{\partial t} + \nabla(\rho \nabla K) = \nabla\left[\frac{\mu_{eff}}{\sigma_k}\nabla k\right] + P_k - \rho\epsilon \tag{4}$$

$$\frac{\partial(\rho\epsilon)}{\partial t} + \nabla(\rho\nabla\epsilon) = \nabla\left[\frac{\mu_{eff}}{\sigma_\epsilon}\nabla\epsilon\right] + \frac{\epsilon}{k}\left[C_1 P_k - \rho C_2 \epsilon\right] \tag{5}$$

where

k	– is the turbulence kinetic energy;
ϵ	– is the dissipation rate of turbulence kinetic energy;
$C_1, C_2, \sigma_\epsilon, \sigma_k$	– are model constants; and
P_k	– is the volumetric production rate of k, defined by

$$P_k = \mu_t\left[\frac{\partial V_i}{\partial x_j} + \frac{\partial V_j}{\partial x_i}\right]\frac{\partial V_i}{\partial x_j}.$$

When single–phase three–dimensional flow is under consideration, the dependent variables that require solution are

p	– the static pressure;
u	– the horizontal–direction velocity component;
v	– the vertical–direction velocity component;
w	– the axial–direction velocity component;
k, ϵ	– turbulence quantities (described above);
h	– the static enthalpy;

C1 – the mass fraction of engine gases.

3.1.2 General Form of the Governing Equations

The governing equation for each dependent variable (φ) can be reduced to a single general form which can be succinctly represented in vector notation as

$$\frac{\partial(\rho\varphi)}{\partial t} + \nabla . \left[\rho \, \nabla\varphi + J_{\varphi}\right] = S_{\varphi} \tag{6}$$

The source term S_{φ} includes both sources and sinks of dependent variable (φ) plus any other terms beyond those which appear on the left–hand side of the equation.

Because of the generality of the SFV code, a standard gradient–diffusion law is provided for the diffusion flux, namely

$$J_{\varphi} = -\Gamma_{\varphi}\nabla\varphi \tag{7}$$

Where diffusion of the conserved quantities does not follow the standard form of Equation 7, compensating terms are added to the source term S_{σ}. In that equation, Γ_{φ} is the exchange coefficient for the variable φ, defined as

$$\Gamma_{\varphi} = \frac{\mu_{eff}}{\sigma_{\varphi,eff}} \tag{8}$$

where $\sigma_{\varphi,eff}$ is the effective Prandtl/Schmidt number governing diffusive transport of the variable φ.

The exception to the general form given by Equation 6 is the continuity (mass conservation) equation. The pressure, p, has been classified as a dependent variable, but does not appear as the subject of a transport equation. Instead, the pressure is associated with the continuity equation which can be manipulated to derive a pressure–correction equation. SFV employs a variant of the SIMPLE pressure–correction procedure [4]. The transport coefficient Γ_φ and source term S_φ are provided for each dependent variable in Table 2.

TABLE 2. Transport Equations and Turbulence Model Constraints

	φ	Γ_φ	S_φ
Continuity Momentum:	1	0	0
x–direction	u	μ_{eff}	$-\frac{\partial p}{\partial x}$
y–direction	v	μ_{eff}	$-\frac{\partial p}{\partial y}$
z–direction	w	μ_{eff}	$-\frac{\partial p}{\partial z}$
Kinetic Energy	k	$\frac{\mu_t}{\sigma_k}$	$P_k - \rho\epsilon$
Dissipation Rate	ϵ	$\frac{\mu_t}{\sigma_\epsilon}$	$\frac{\epsilon}{k}(C_1 P_k - C_2 \rho\epsilon)$
Enthalpy	h	$(\frac{\mu}{\sigma} + \frac{\mu_t}{\sigma_t})$	0
Concentration	C	$(\frac{\mu}{\sigma} + \frac{\mu_t}{\sigma_t})$	0

$C\mu = 0.09$, $C_1 = 1.44$, $C_2 = 1.92$, $\sigma_k = 1.0$, $\sigma_\epsilon = 1.314$

3.1.3 General Form of the Finite–Domain Equations

Integration of the generalized conservation equation (6) yields a finite–difference analog for each dependent variable φ, for each control cell into which the flow geometry is divided. The integration process is carried out, and the terms in the finite–domain equations are

assembled in the manner described in many standard texts on numerical analysis [5]. For each simulation performed, the SFV program will assemble and then solve a set of simultaneous coupled equations of the general form (for point P)

$$a_P\varphi_P = a_E\varphi_E + a_W\varphi_W + N\varphi_N + a_S\varphi_S + a_H\varphi_H + a_L\varphi_L + S_\varphi \tag{9}$$

where the subscripts E, W, N, S, H, and L represent the neighbor points in space. The coefficients, a_N and others, represent the effects of convection and diffusion. S_φ is the linearized source term. The central coefficient a_P is defined as

$$a_P = a_E + a_W + a_N + a_S + a_H + a_L \tag{10}$$

The set of equations (9) are assembled and solved for each dependent variable in sequence. The solution of the flow field then proceeds iteratively to steady-state.

3.1.4 Auxiliary Relations

To complete the mathematical formulation of a problem the specification of additional information is required: namely, fluid properties and boundary conditions.

SFV allows the arbitrary specification of fluid properties. The models resulting from this study will employ fluid properties typical of the conditions present in the Naval Air Station, Sigonella, Italy test cells. SFV also allows several different types of flow and thermal boundary conditions. These will be appropriately chosen so as not to sacrifice the fidelity of the models and also to allow for easy modification in the course of the parametric studies.

4. NUMERICAL DETAILS AND PHYSICAL MODELS

4.1 Introduction

This section of the report describes the various aspects of the modeling method necessary to simulate the flow and heat transfer inside the test facility. First, the computational grid employed in this simulation is described. Next, the physical properties are defined. This is followed by a discussion of the boundary conditions used in the computational model.

4.2 Grid Description

The first step in setting up a numerical model is the specification of the computational grid. For this study a three dimensional Body–Fitted Coordinate (BFC) grid was employed. The grid for this modeling effort is quite complex. In order to simplify the grid generation, a procedure has been developed to allow for relatively easy modifications of the test cell configuration.

In this method the user specifies various geometric quantities. The standard input files then create a set of data files for the EasyMesh [6] program which is a two dimensional BFC package. The user must then run this grid generation package to create a set of two dimensional grids. The standard input files are then re–executed to produce the final grid by stacking, blending, and rotation of the two dimensional grid planes.

The final computational grid for this study is shown in Figure 1. This grid has 34 cells across the test bay, 36 cells in the vertical direction, and 75 computational cells in the axial direction. The grid is clustered in areas of geometric changes, areas of expected high shearing action, and areas of heat transfer. In Figures 2 through 4 outlines of the internal components are presented.

4.3 Properties of the Fluid

The properties of the gases mixture inside the test cell were calculated based on the local conditions. The local heat capacity was computed using the gas mass fraction and the cell

temperature using JANNAF [7] polynomial expressions. The gas density was calculated using the ideal gas law (requiring local pressure, molecular weight, and temperature). The laminar kinematic viscosity was assumed to have a constant value of 1.08×10^{-4} ft^2/s (1.0×10^{-5} m^2/s). The turbulent viscosity, which tends to dominate, was calculated from the local turbulence quantities.

Heat transfer through the augmenter tube wall required the specification of the material thermal conductivity. Inside the room the thermal conductivity of mineral fiber with a value of 0.22 Btu/ft–s–R (0.38 W/m–K), was used. In the chimney the conductivity of steel was used. The value of steel was taken to be 26 Btu/ft–s–R (45 W/m–K).

4.4 Boundary Conditions

For this model, the boundary conditions requiring specifications were

- the engine,
- the propeller,
- inlet into test cell,
- outlets of test cell and chimney,
- wall boundaries, and
- heat transfer through the augmenter wall.

These boundary conditions and sources are discussed in the following sections.

4.4.1 Engine

The engine is modeled as a hollow cylindrical shape with a variable cross sectional area. Inside this region a plate of computational faces were blocked. On the intake side a mass sink is applied which corresponded to the exit mass flow minus the fuel intake.

On the exhaust side a mass source was applied to account for the engine mass flow. Other boundary conditions applied the appropriate sources for the momentum, enthalpy, turbulence, and concentrations. The exit turbulence quantities were calculated assuming a

15 per cent turbulence intensity from the following equations:

$$k = 1/2\,(vI_t) \tag{11}$$

$$\epsilon = 0.1643\, k^{1.5}/l_m \tag{12}$$

where

I_t – is the turbulence intensity and

l_m – is the mixing length (based on a gap distance or inlet radius)

4.4.2 Propeller

The propeller was modeled using an actuator disk model, where in all of the net results of the propeller (wash, swirl, turbulence) are modeled as source terms in the momentum and turbulence equations. The three aspects of the propeller which were modeled were thrust (axial flow generation), swirl (swirling flow generation) and turbulence (turbulence generation by the whirling blades). The actual methodology of modeling each of these three physical characteristics is now detailed.

4.4.2.1 Thrust

The thrust of the propeller was modeled as an momentum source in the axial–direction momentum equation. The actual expressions used were supplied by Hamilton Standard Corporation, the manufacturer of the propeller being modeled. The performance of a propeller is characterized by correlating the ratio of the thrust coefficient (C_T) versus the power coefficient (C_P). These coefficients are defined as:

$$C_P = P/\rho n^3 D^5 \tag{13}$$

with

P – power absorbed by propeller,

ρ – reference fluid density,

n – rotational speed of propeller, and

D — diameter of propeller:

$$C_T = T/\rho n^2 D^4 \tag{14}$$

with

T — thrust produced by the propeller.

All quantities are assumed to be in consistent units. The propeller is characterized by a plot of C_T/C_P versus C_P. This plot was also supplied by Hamilton Standard (see Appendix B). Thus, the user of the numerical model supplies shaft horse power SHP (which gives power), RPM (which gives n), and prop diameter. The code converts all of these to MKS units and evaluates a C_P. The prop characteristic curve is coded into GROUND by data statements and the code interpolates as required to generate C_T/C_p. With the evaluation of C_T the thrust is calculated from Equation 14.

The average axial induced velocity of the propeller is given as

$$W_p = \sqrt{\frac{T}{2\rho A_p}} \tag{15}$$

with

A_p — area of whirling prop ($\pi D^2/4$).

The code computes W_p as a check on prop model performance only, it is not otherwise needed. The thrust per unit–mass–of–air within the envelope of the propeller is given as:

$$T/\text{mass} = T/\rho Vd = T/\rho A_p \delta_p = 2W_p^2/\delta_p \tag{16}$$

where δ_p is the thickness of the grid cells in the axial direction where the prop is located. This is how the source is actually implemented in the code for numerical consistency with the rest of the discretized equations.

4.4.2.2 <u>Swirl</u>

The integrated swirl generated consumes a certain percentage of engine power. This percentage is calculated by having the user specify the percentage of total power which is assumed to be wasted generating turbulences (%k). The percentage used to generate thrust (%w) is known from the thrust calculation as follows. The power used to generate thrust, P_T is given as:

$$P_T = TW_p \tag{17}$$

Then the percent of power used in thrust is $P_T/P*100$.

It is assumed that all engine power absorbed by the prop is used to do one of three things, namely thrust, swirl, and turbulence, and knowing %w, and %k, the percentage used for swirl (%u) is calculated. The power used by the swirl (P_u) is %u*P. It is assumed that the power used in swirl generation is distributed evenly over the face of the propeller. This power is equal to the torque used in swirl generation times the angular speed of the prop:

$$P_u = Tn \tag{18}$$

This torque is equal to the local force of the prop on the air in the direction of swirl times the distance from the center of the prop to the location under consideration.

$$T = Fr \tag{19}$$

This force is assumed to be constant in r; thus

$$P_u = Frn \tag{20}$$

or F, the magnitude of the propeller on the air locally, is given as

$$|\vec{F}| = P_u / r\, n \tag{21}$$

The code computes local unit vectors for the prop swirl force $\vec{F}$, then supplies $\vec{F} \cdot \hat{e}_u$ as the local swirl source for the u and v momentum equations, where $\hat{e}_u$ is the local unit vector for the u and v momentum equations, available from the code.

4.4.2.3 Turbulence

The turbulence of the propeller is assumed to be proportional to local prop speed squared:

$$S = Cv^2 \tag{22}$$

The power used to generate, turbulence, (P_k = %k*P) is assumed to be distributed over the radius of the propeller such that the integrated power dissipated is equal to P_k. Thus

$$P_k = \int_0^{2\pi} \int_0^{R} C\, v^2\, r\, dr\, d\theta \tag{23}$$

The constant C is evaluated by completing the integral and solving for C to give

$$C = \frac{64\ P_k}{2\pi\ n^2\ D^4} \tag{24}$$

and the local turbulence sources are evaluated as

$$S_k = C(rn)^2 \text{ (turbulence kinetic energy)} \tag{25}$$

and

$$S_\epsilon = S_k \frac{\epsilon}{k} \text{ (dissipation of turbulence kinetic energy).} \tag{26}$$

4.4.3 Inlet

At the inlet a fixed pressure was applied which corresponded to the ambient pressure. From this and the other boundary conditions the mass flow was computed. Since this mass flow varies during the solution process, the momentum source was continually updated according to the computed inflow. The enthalpy and concentrations were considered to be ambient. A turbulence intensity of 2 per cent was used to calculated the incoming turbulence values. A constant velocity of 3.3 ft/s (1.0 m/s) was used in the turbulence calculations.

The baffles located at the inlet and chimney were not directly modeled. These effects were modeled by calculating the momentum loss through the baffles from

$$\Delta p = 0.5\ K \rho v^2$$

where

K is a loss coefficient.

Supplied with pressure drop across the baffles along with gas density and velocity an accurate determination of K is possible. Pressure data from the turboshaft test cell indicated a k–loss factor of 0.2. Because the pressure drop in the test cell was under predicted a value of 1.0 was used in this simulation.

4.4.4 Outlet

At the two 'outlets' a fixed pressure was used. If no recirculation occurs this is all that is required. However, in the chimney, the possibility of recirculation at the exit was extremely great. Momentum and energy sources were supplied if and when recirculation at the exit plane was calculated. The momentum source was based on the amount of mass being brought into the domain and the energy source was based on ambient gas conditions.

4.4.5 Wall Boundaries

At the wall boundaries the wall function approach [3] was used to account for momentum losses. This condition is not applied directly to the wall but rather at a point outside the viscous sublayer, where the logarithmic law of wall prevails. This is where turbulence is assumed to be in local equilibrium (i.e., the generation and dissipation of turbulence energy

are equal). Heat transfer was not considered except through the augmenter tube wall.

4.4.6 Heat Transfer through Augmenter Tube

To compute the heat transfer through the augmenter tube, a composite heat transfer treatment was used. Surface heat transfer coefficients on both surfaces (inside and outside) were calculated using the standard SFV wall friction treatment. The overall heat transfer conductance between the fluid cell adjacent to the inner wall and the one adjacent to the outer wall was evaluated using these two surface coefficients (in the form of a heat transfer resistance) and the assumed thermal resistance through the tube wall. This last resistance was evaluated using a conductivity of insulation and thickness of insulation provided by the Navy. In the chimney, where there was no insulation, the conductivity and thickness of the augmenter tube wall was used for the solid contribution to the overall heat transfer conductions.

5. DISCUSSION OF RESULTS

This report presents the results obtained from one computational run. At the time of this report there were no experimental data for comparison purposes.

5.1 Velocity

The velocity vectors are presented in Figures 5 through 10. A cross sectional view to the vectors are shown in Figure 5. This plot provides a good overall display of the major flow patterns within the calculation domain. Two plots of velocity around the propeller are shown in Figures 6 and 7. The first one again shows a side view. In this plot the air movement through the orifice and propeller is presented. The effect of the reduction gear is noted as the flow of air is forced around the unit. Large recirculation zones at the top and the bottom of the test cell downstream of the prop are also shown. In Figure 7 (x–view) the velocity vectors one computational cell down stream of the propeller are plotted. The highest magnitude of velocity is located at the center of the prop. This is from the assumption that a free vortex type propeller was used as noted in Section 4.4.2

Figure 8 shows the velocity vectors at the end of the engine and start of the augmenter tube. There is entrainment of surrounding gases into the tube at this cross section. A later temperature plot will indicate if there is any flow coming out of the augmenter opening. The maximum velocity of the engine is 721 ft/s (218 m/s). The velocity in the chimney section is shown in Figure 9. The augmenter tube is circular down the full length. The axial momentum propels the gases inside the augmenter tube to the aft side of the bend section. The gases do not have enough time to spread the entire cross section of the chimney. This causes a recirculation zone at the exit of the calculation.

The final vector plot (Figure 10) shows a top side view of the velocity vectors entering into the test cell. The entrance velocity is approximately 23 ft/s (7 m/s). The inlet section of the turboprop facility covers the full width and height of the test bay.

5.2 Temperature

Temperature plots are provided in Figures 11 through 15. The overall temperature field is shown in Figure 11. In this model the engine is the only heat source. The temperature around the exit of the engine is shown in Figure 12. The exit temperature of the engine is 1100 F (600 C). This temperature reduces as the engine gases mix with entrained air and

propeller wash in the augmenter tube as shown in Figure 12. The plot of temperature at the exit of the augmenter tube is given in Figure 13. At this point the temperature ranges from 602 F to 626 F (317 C to 330 C). The average temperature is approximately 615 F (324 C). The narrow range in temperature indicates a good level of mixing in the augmenter tube. In Figure 14 the temperature at the exit of the chimney section is plotted. A small zone of recirculation is noted by a region where the temperature is 83 F (28 C) which is close to ambient conditions of 77 F (25 C). During the iterative solution process, intermediate unconverged results indicated that a flow was exiting the entrance of the augmenter tube. In Figure 15 a plot of the temperature in the first cell in the augmenter lip is provided. It shows no recirculation of hot gases out of the lip region.

5.3 Other Quantities

In Figure 16 the pressure field located approximately 2 ft (0.6 m) from the right side looking into the cell is displayed. In this plane there is both cell depression and cell pressurization. The values range from −1.2 to 1.4 inches of water.

Figure 17 shows the turbulence kinetic energy (TKE) at the exit of the engine. The largest values occur in the shear region between the top of the engine and the top of the augmenter lip. In Figure 18 the TKE is plotted one cell down stream of the propeller. The largest values occur near the tip of the prop.

There was one global number calculated from this model. This was the pumping ratio of the engine. The calculated value was 1.08.

Heat transfer through the augmenter tube was at the rate of 1.6 BTU/s (1680 J/s). Approximately 55 per cent of this occurred in the uninsulated section of the chimney.

6. CONCLUSION

A 3–dimensional turboprop model of a Naval test cell facility has been developed. This model allows the user to investigate various parameters of interest including variations in geometry, ambient conditions, and boundary conditions. The results of this study show

- an engine pumping ratio of 1.08;
- cell depression and pressurization that range between –1.2 and 1.4" of water;
- an augmenter tube average exit temperature of 615 F; and
- recirculation at the exit of the chimney section.

There were no experimental values to compare with predicted values.

7. REFERENCES

1. "The SFV Beginner's Guide, "Adaptive Report 100, 1992.

2. "The SFV Reference Manual", Adaptive Report 200, 1992.

3. Launder, B. E. and Spalding, D. B., Mathematical Models of Turbulence, Academic Press, 1972.

4. Patankar, S. V. and Spalding, D. B, "A Calculation Procedure for Heat, Mass and Momentum Transfer in Three–Dimensional Parbolic Flows," Int. Journal of Heat and Mass Transfer, pp 1787–1806, 1972.

5. Patankar, S. V., Numerical Flow and Heat Transfer, Hemisphere, New York, 1980.

6. Daley, P. L. and Owens, S. F., "User Manual for Grid Generation Program", CHAM Report 2008/2, November 1988.

7. Gordon, S. and McBride, B., "Computer Program for Calculation of Complex Chemical Equilibrium Compositions", NASA SP–273, 1971.

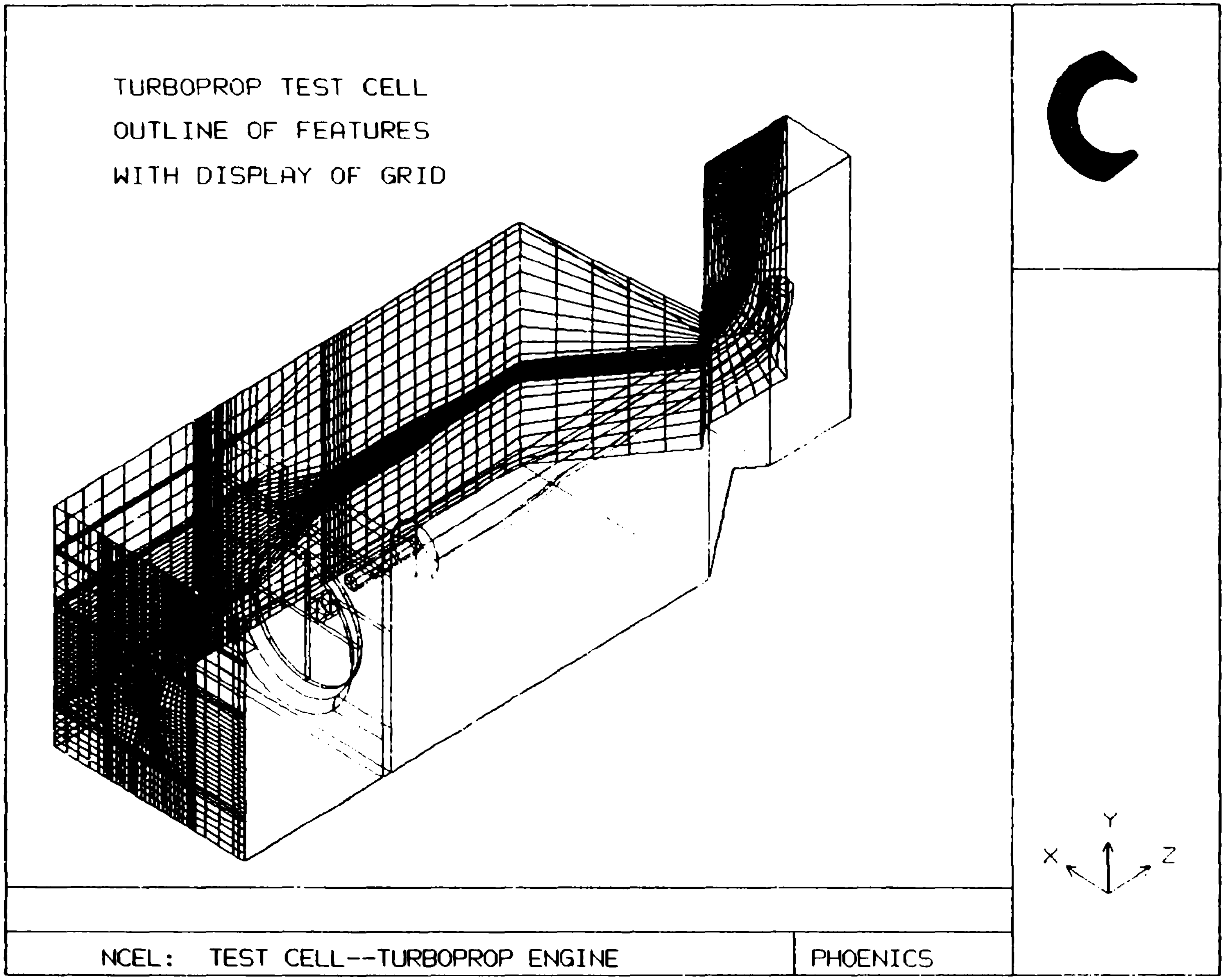


Figure 1 Computational Grid

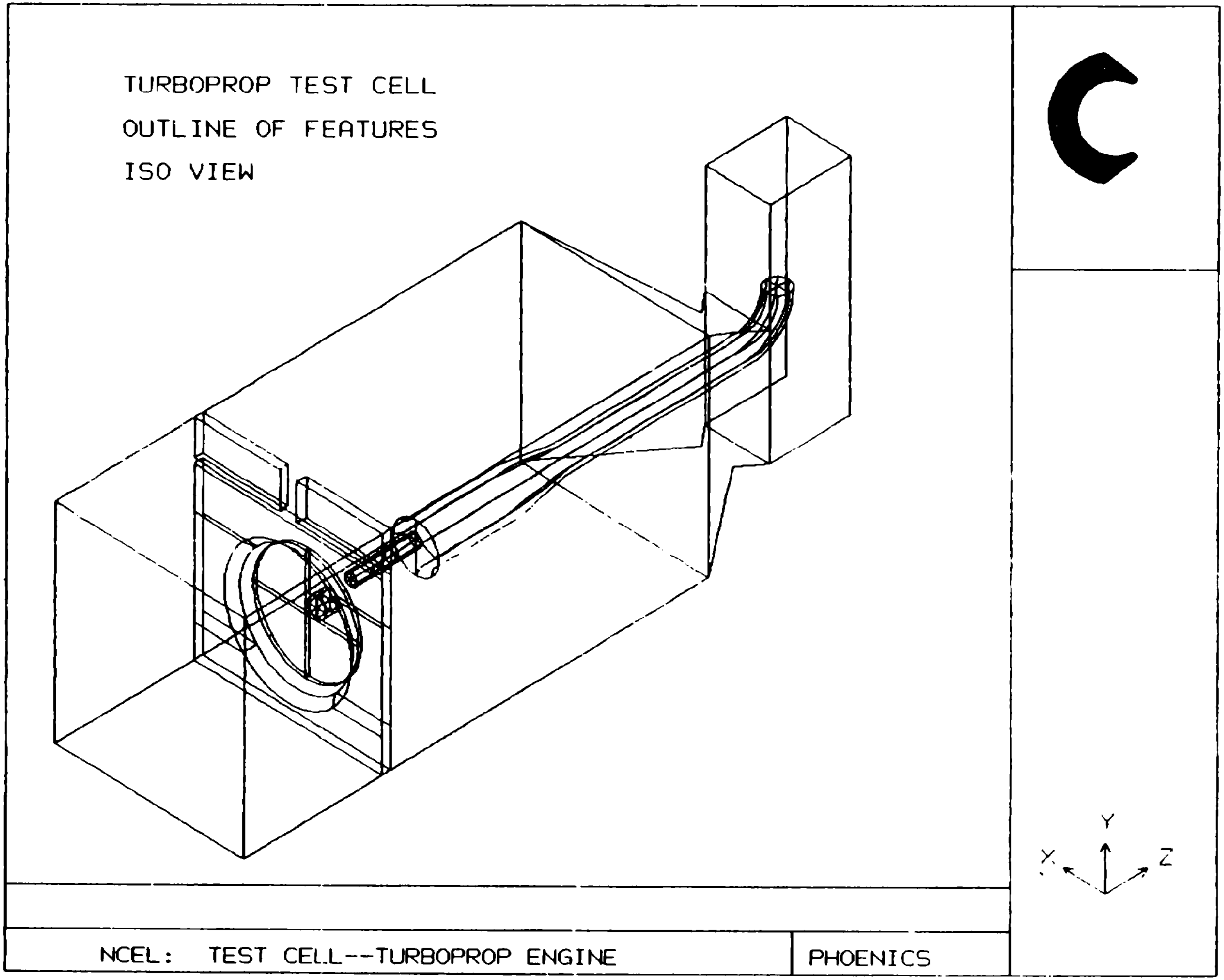


Figure 2 Outline of Features (ISO View)

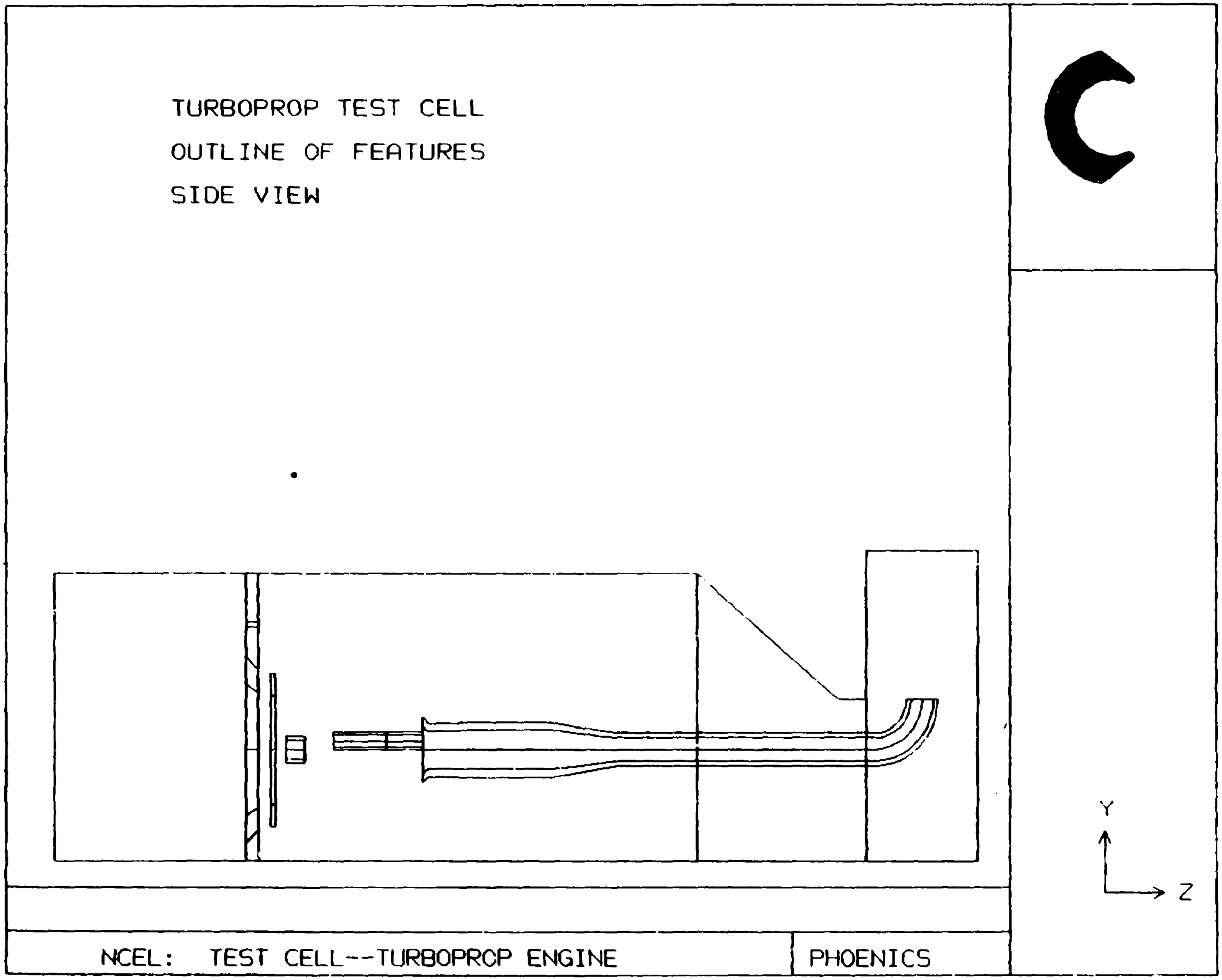


Figure 3 Outline of Features (Side View)

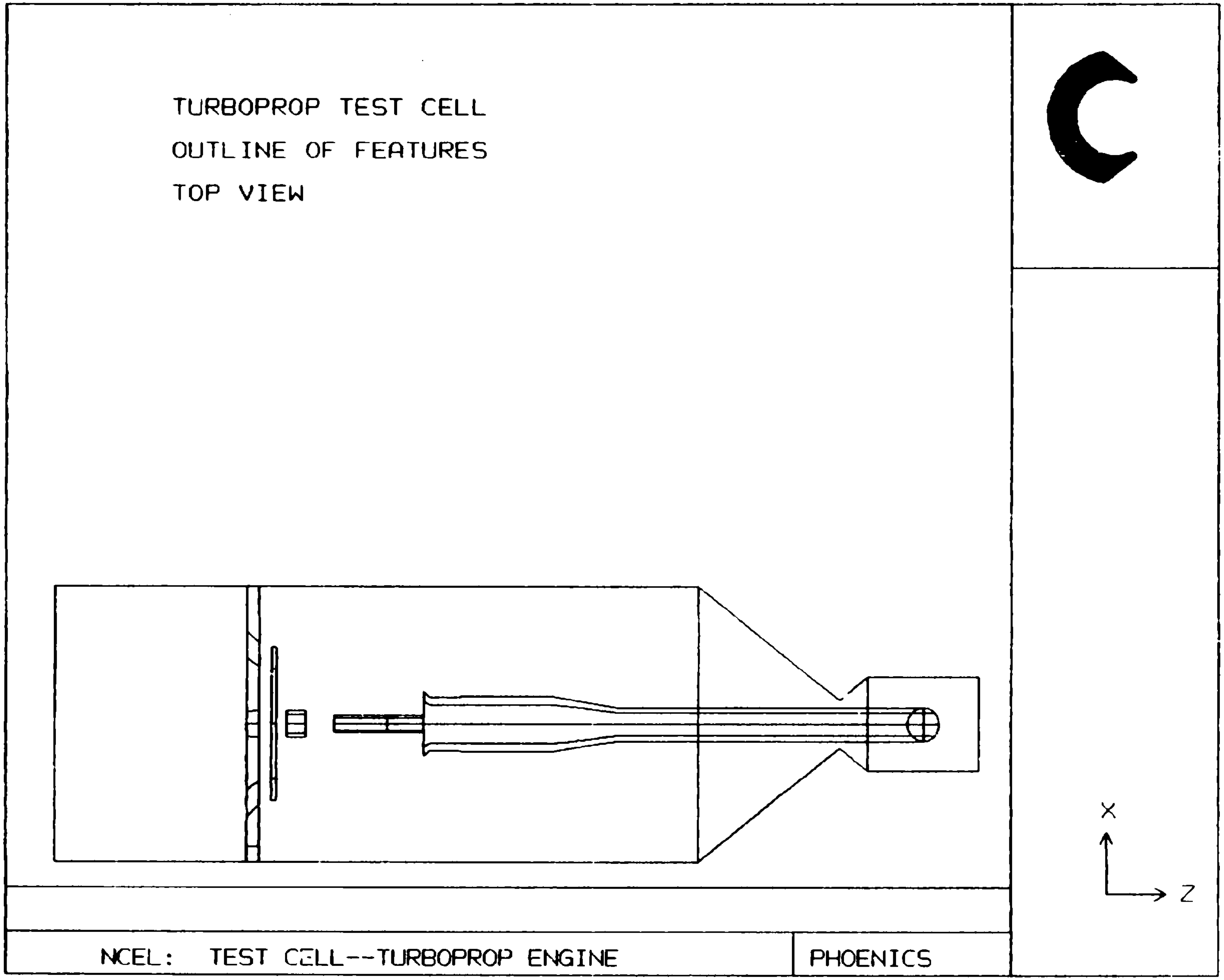


Figure 4 Outline of Features (Top View)

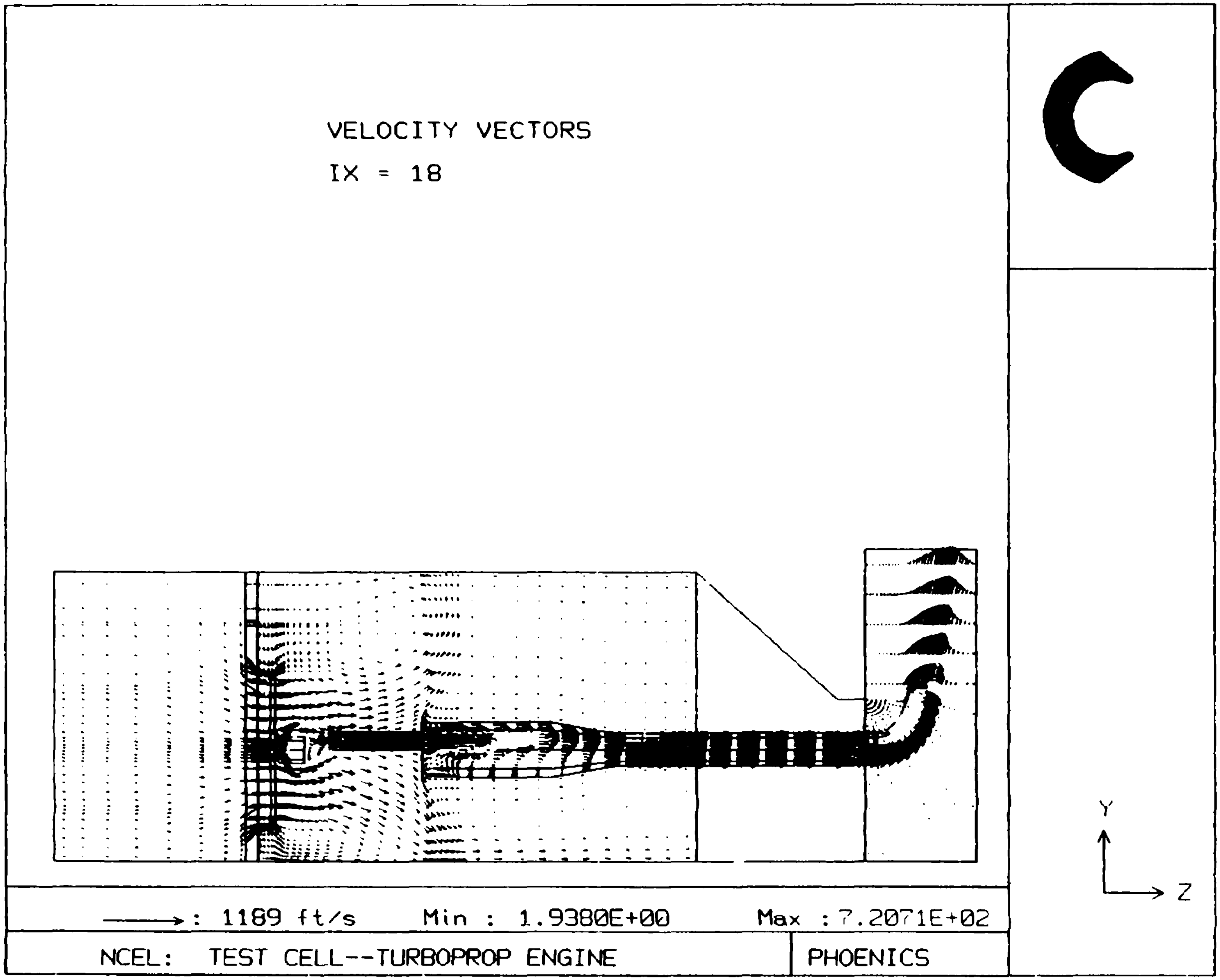


Figure 5 Velocity Vectors (Vertical Plane Through Engine)

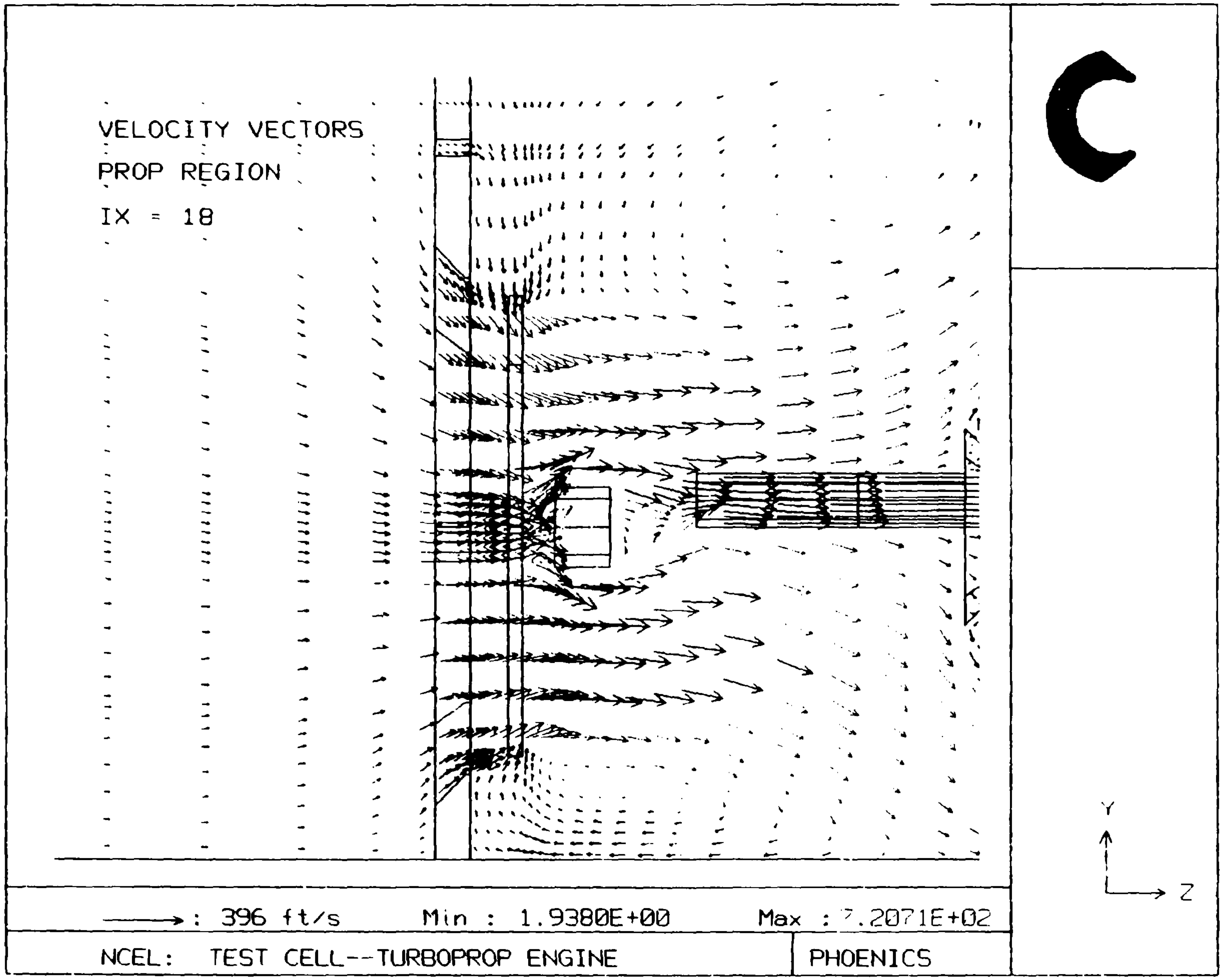


Figure 6 Velocity Vectors Prop Region (Vertical Plane)

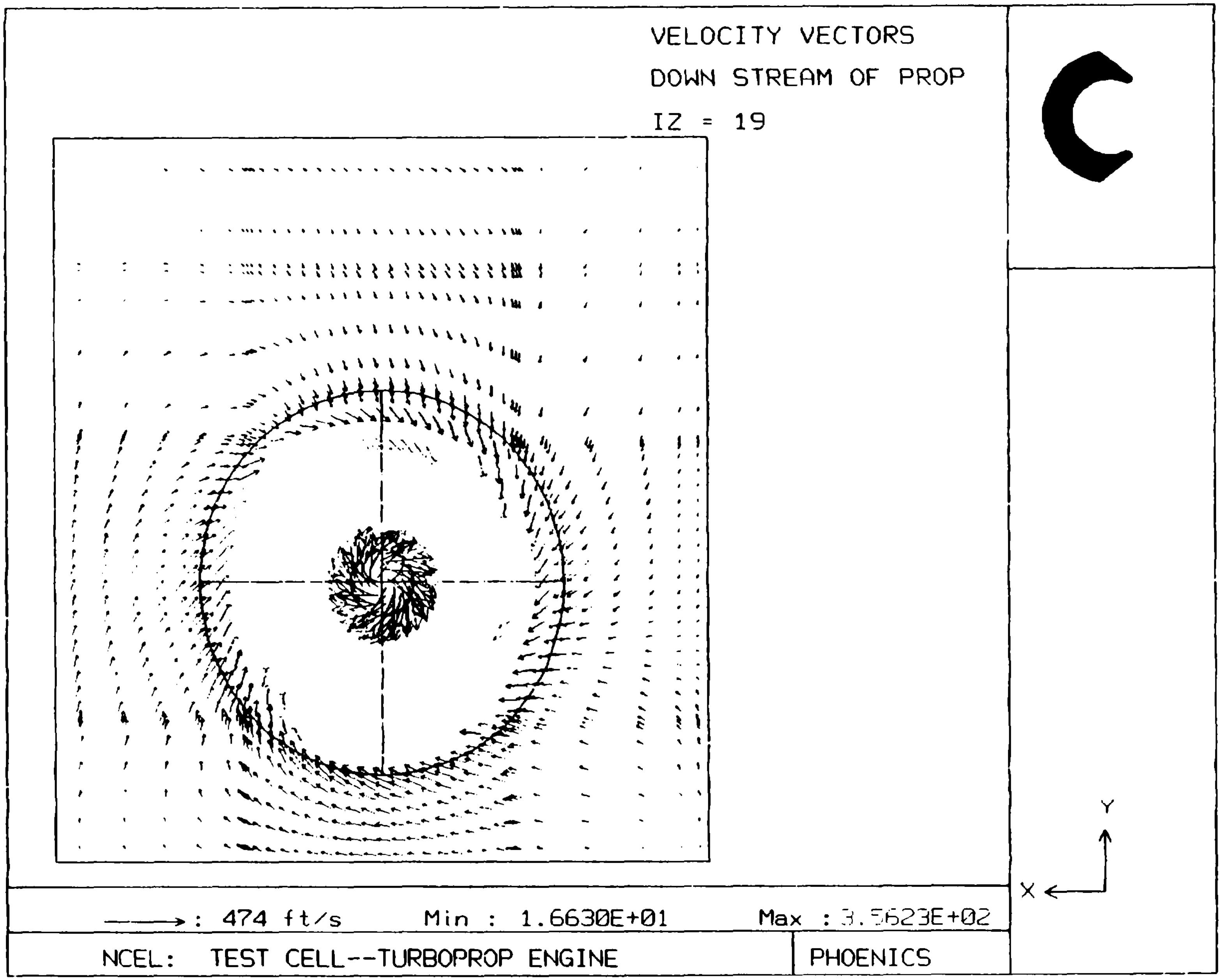


Figure 7 Velocity Vectors Down Stream of Prop (Cross Plane)

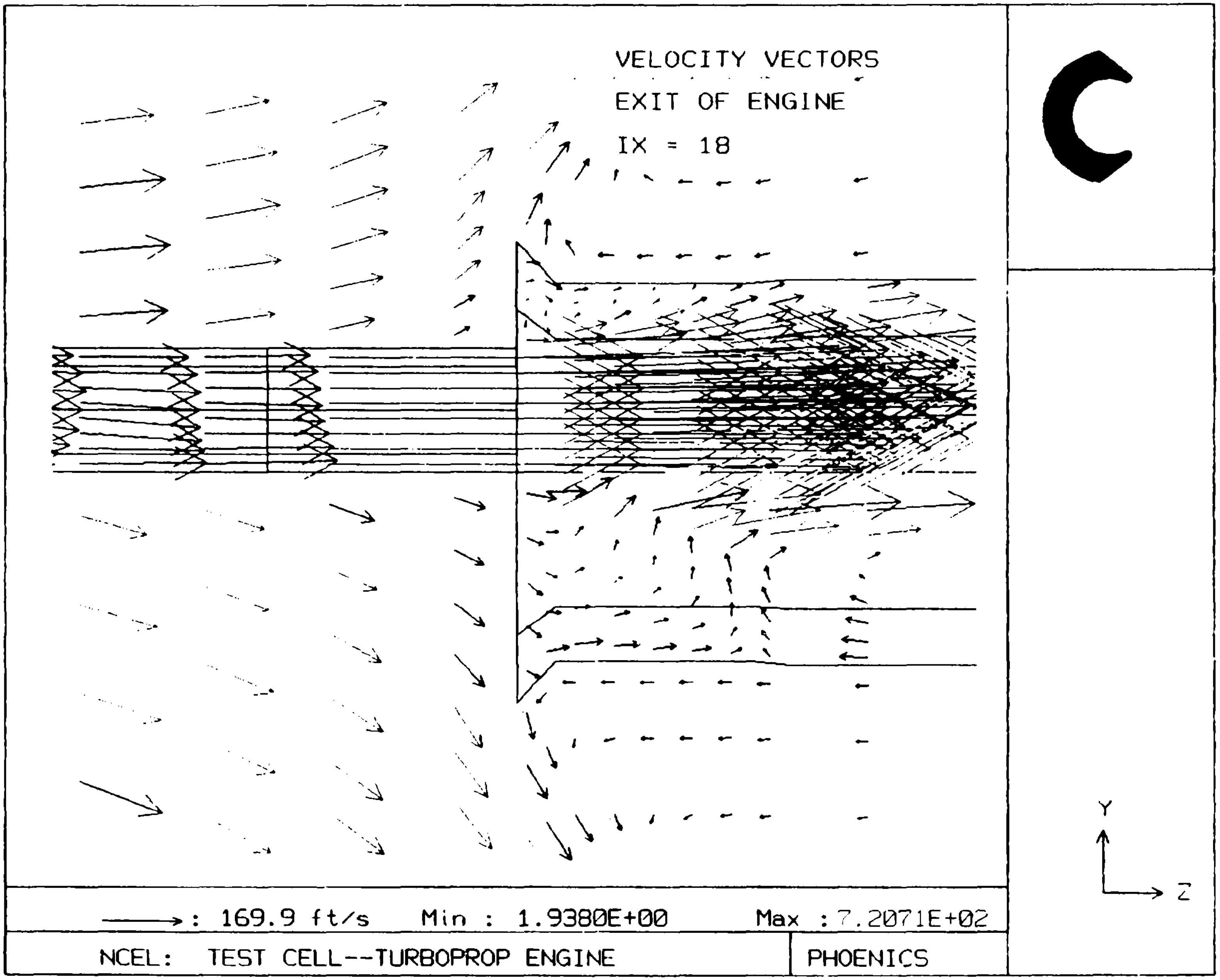


Figure 8 Velocity Vectors Exit of Engine (Vertical Plane Through Engine)

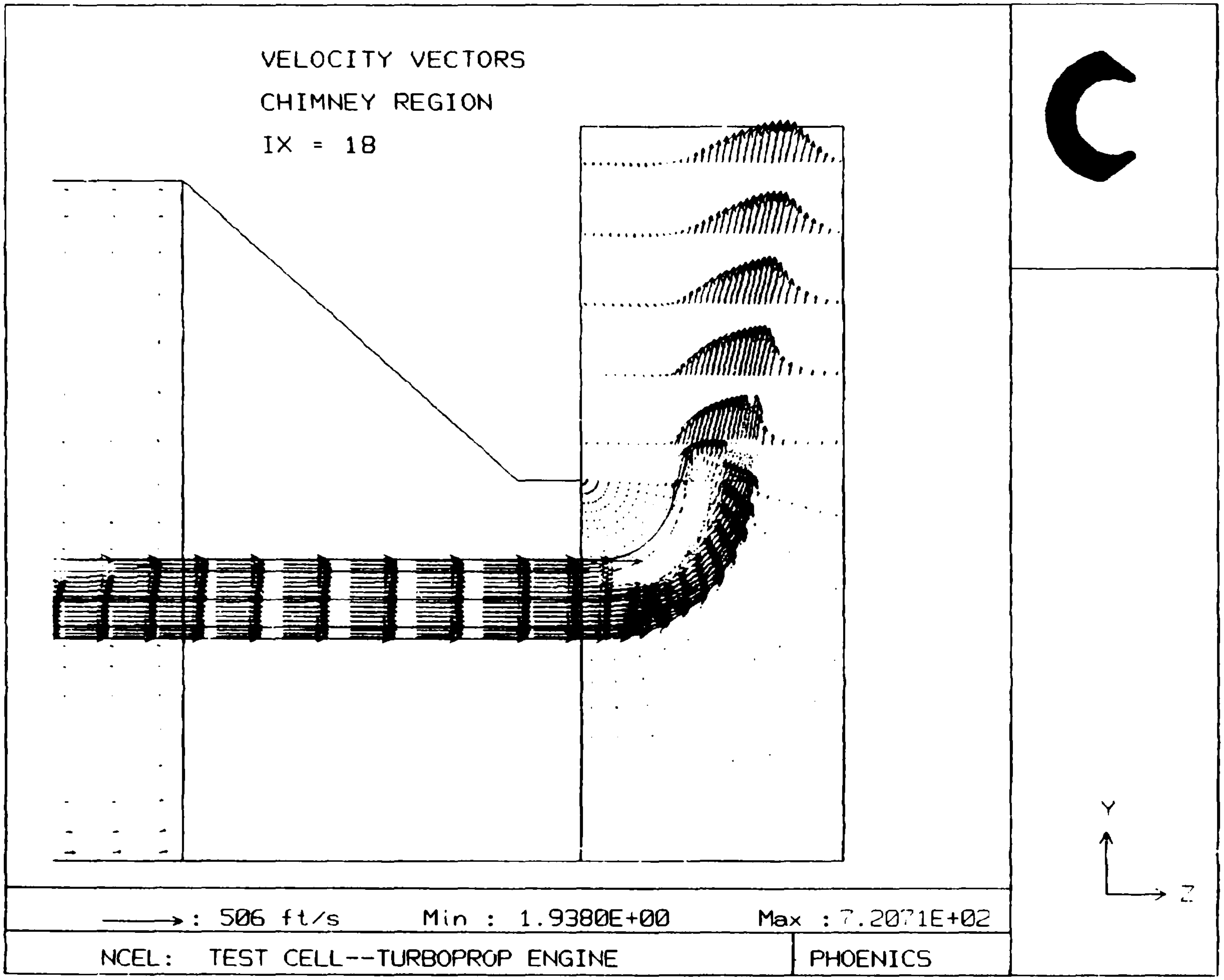


Figure 9 Velocity Vectors Chimney Section (Vertical Plane)

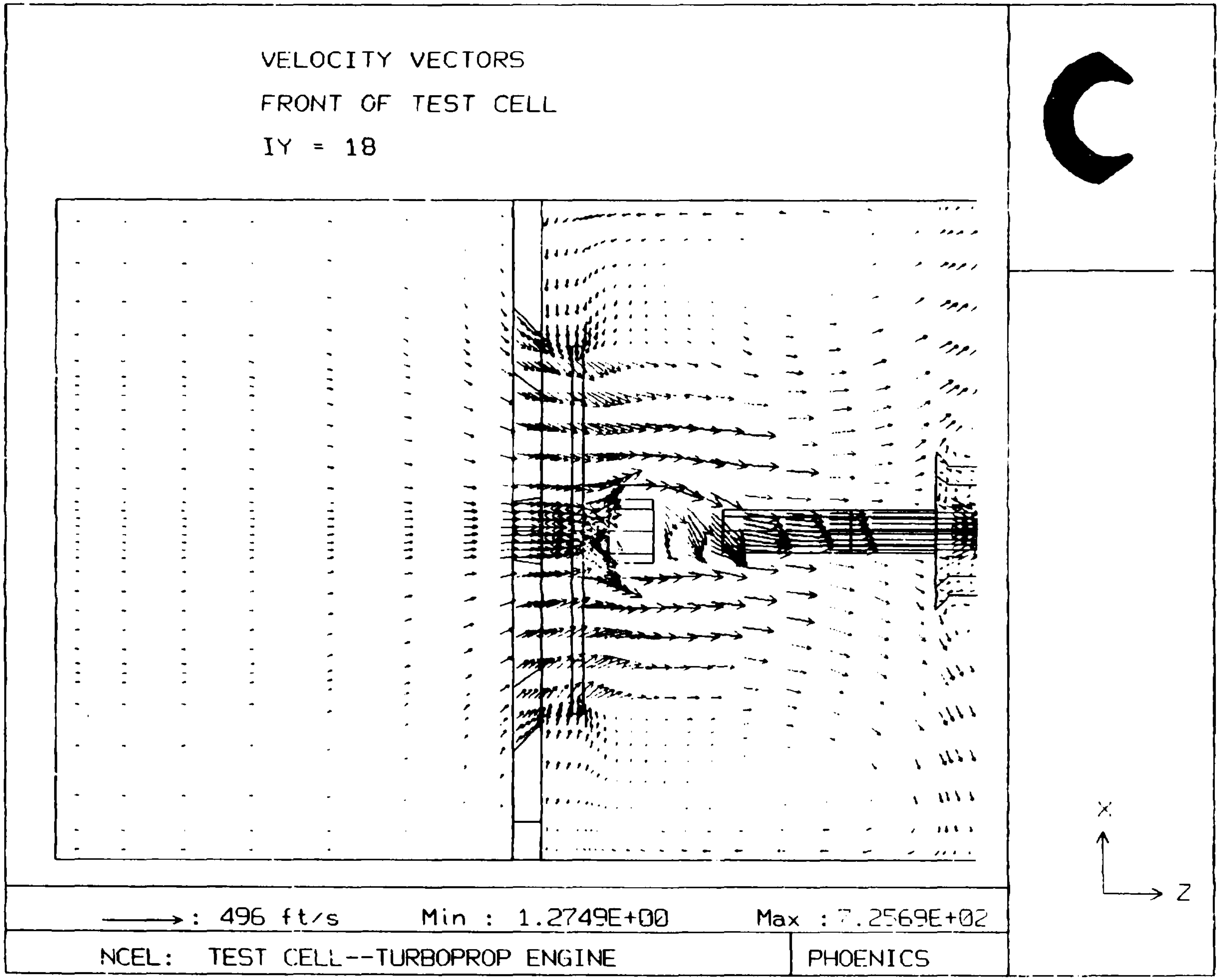


Figure 10 Velocity Vectors Front of Test Cell (Horizontal Plane Through Engine)

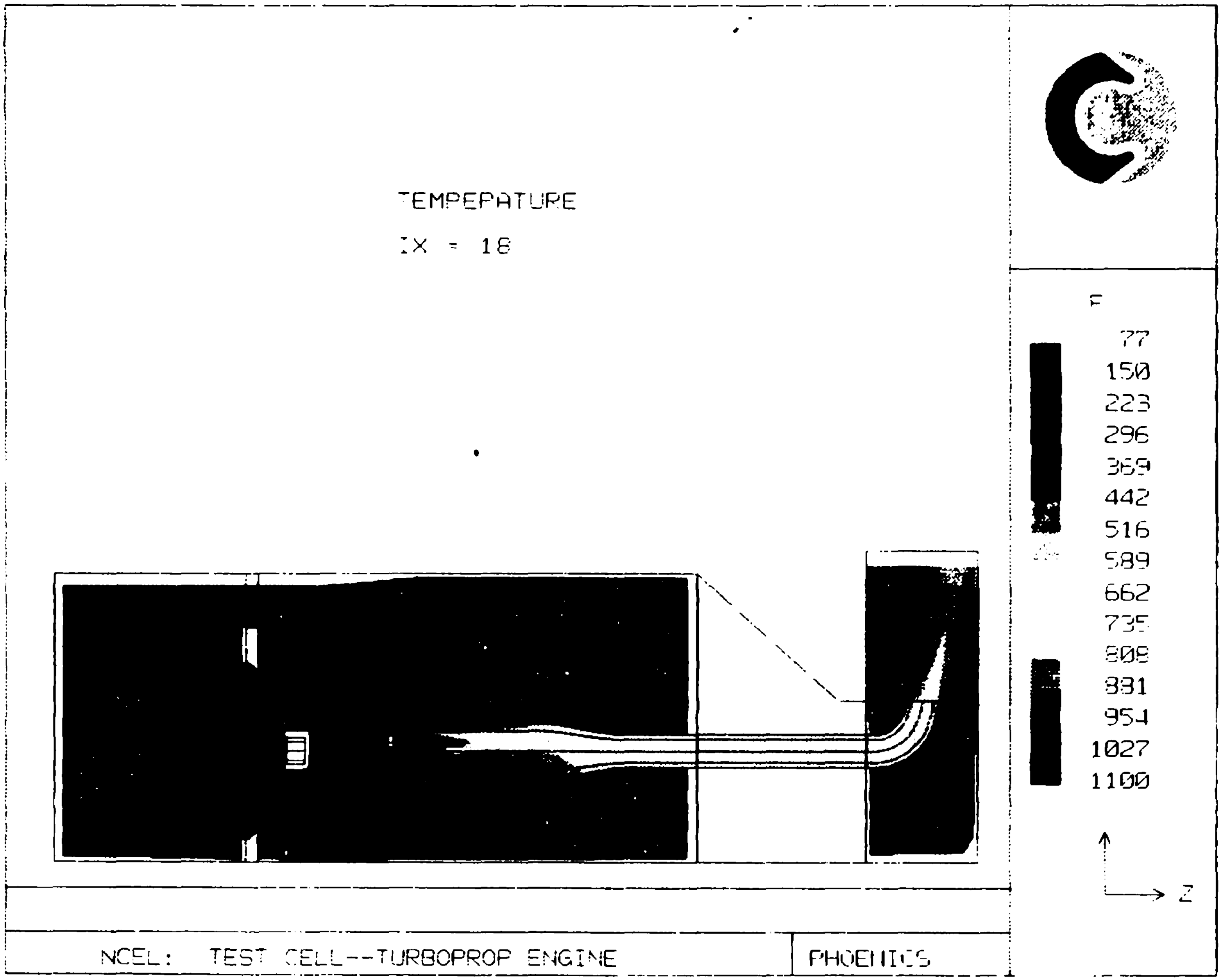


Figure 11 Temperature (Vertical Plane Through Engine)

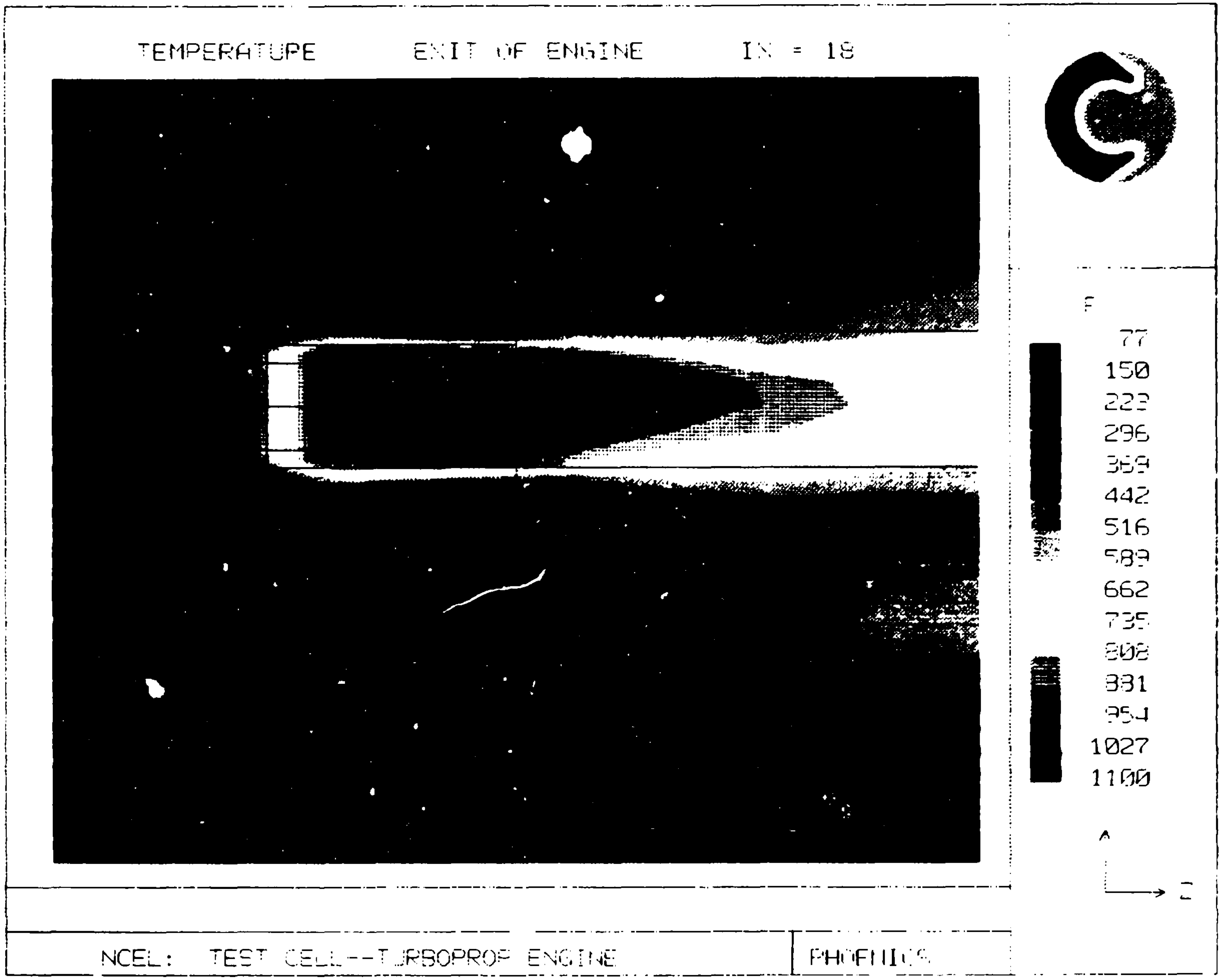


Figure 12 Temperature Exit of Engine (Vertical Plane Through Engine)

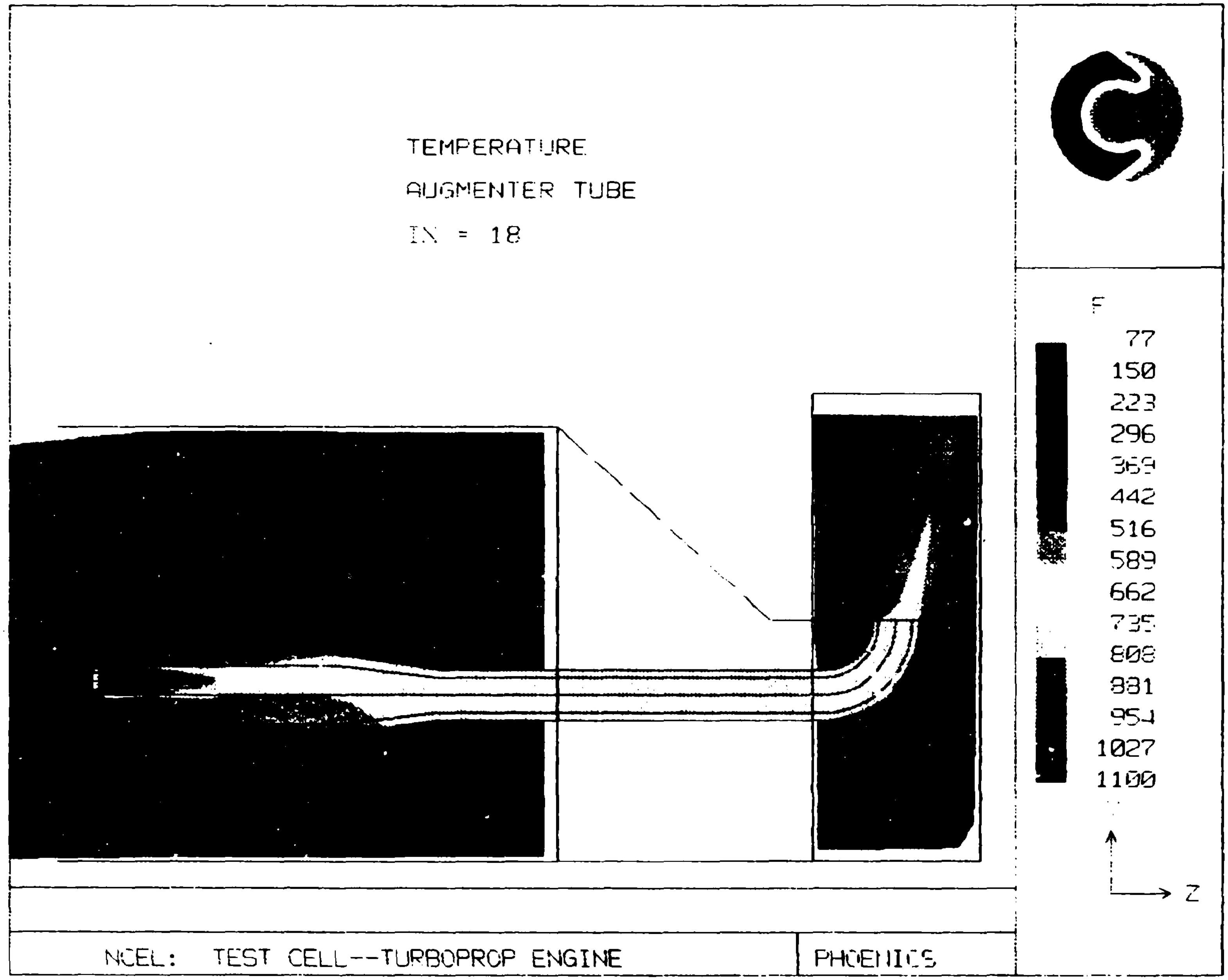


Figure 13 Temperature Augmenter Tube (Vertical Plane)

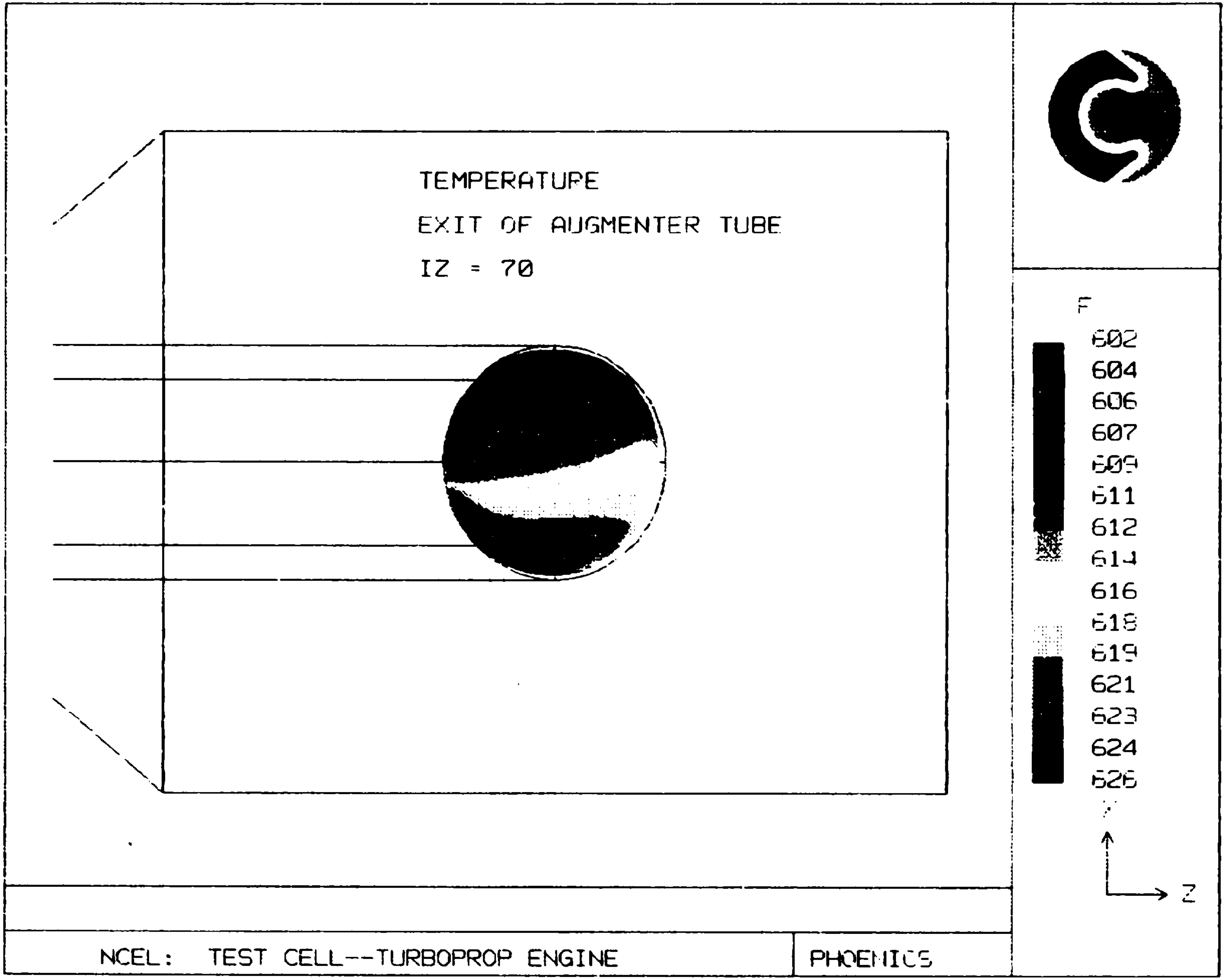


Figure 14 Temperature Exit of Augmenter Tube (Horizontal Plane)

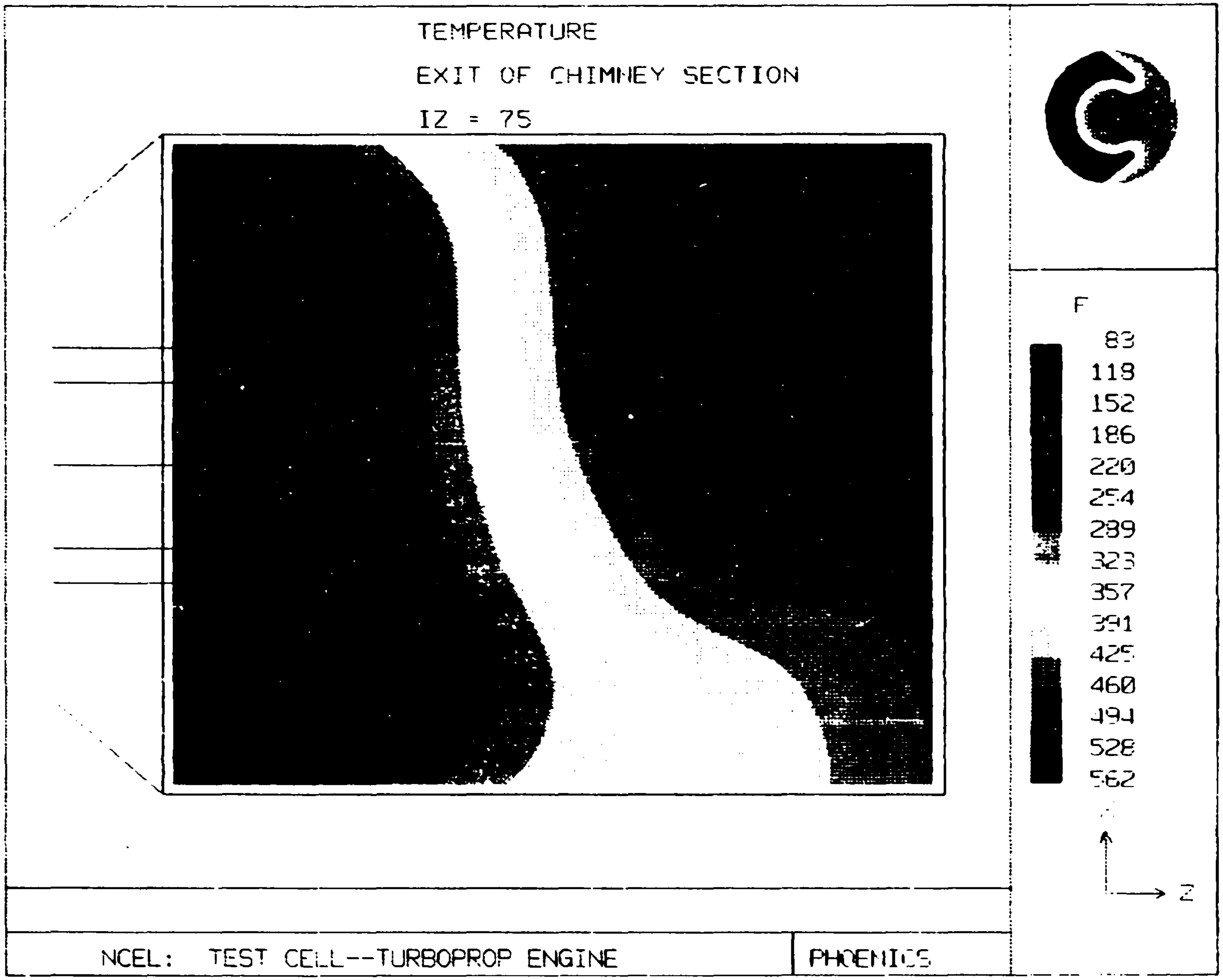


Figure 15 Temperature Exit of Chimney Section (Horizontal Plane)

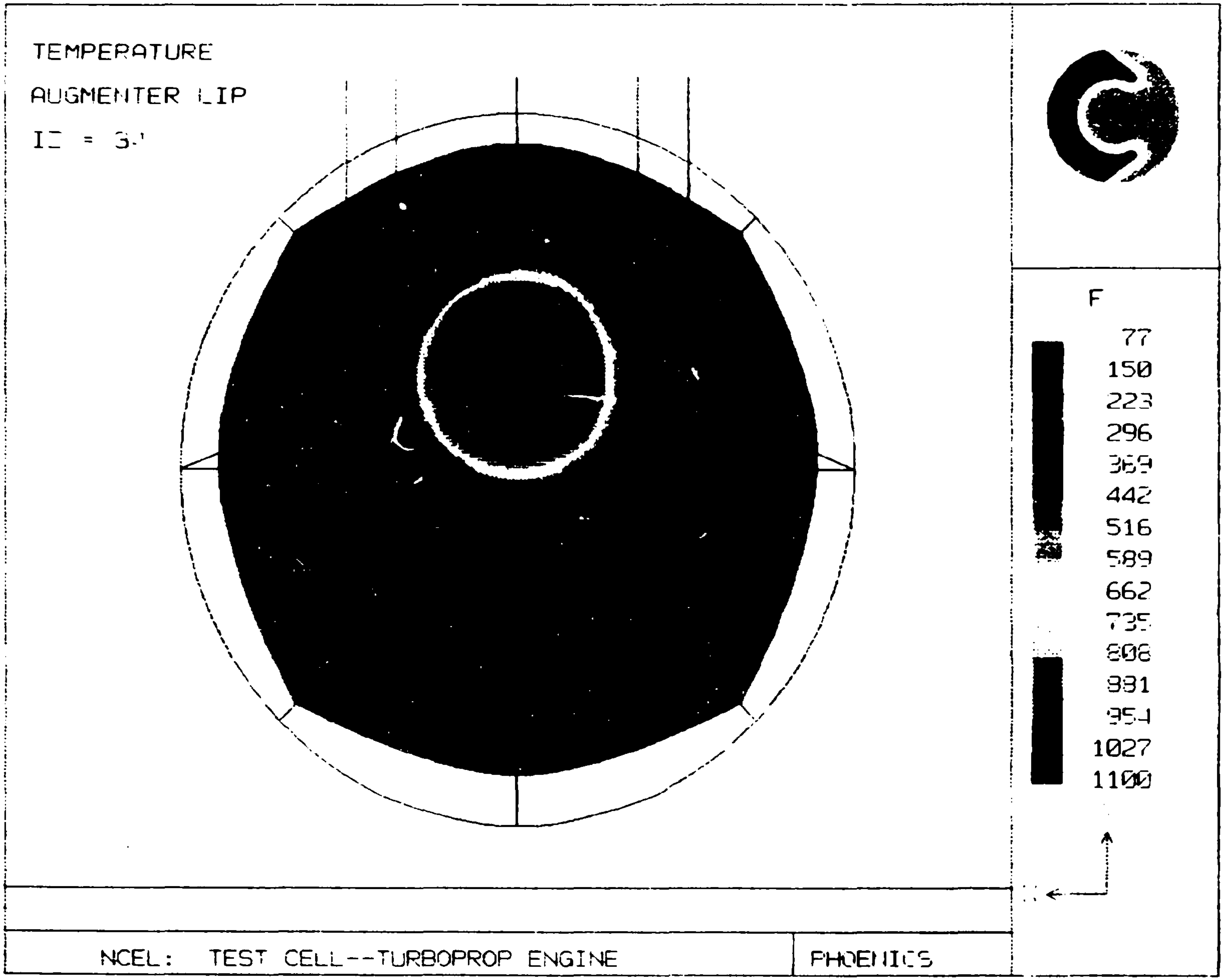


Figure 16 **Temperature Augmenter Lip (Horizontal Plane)**

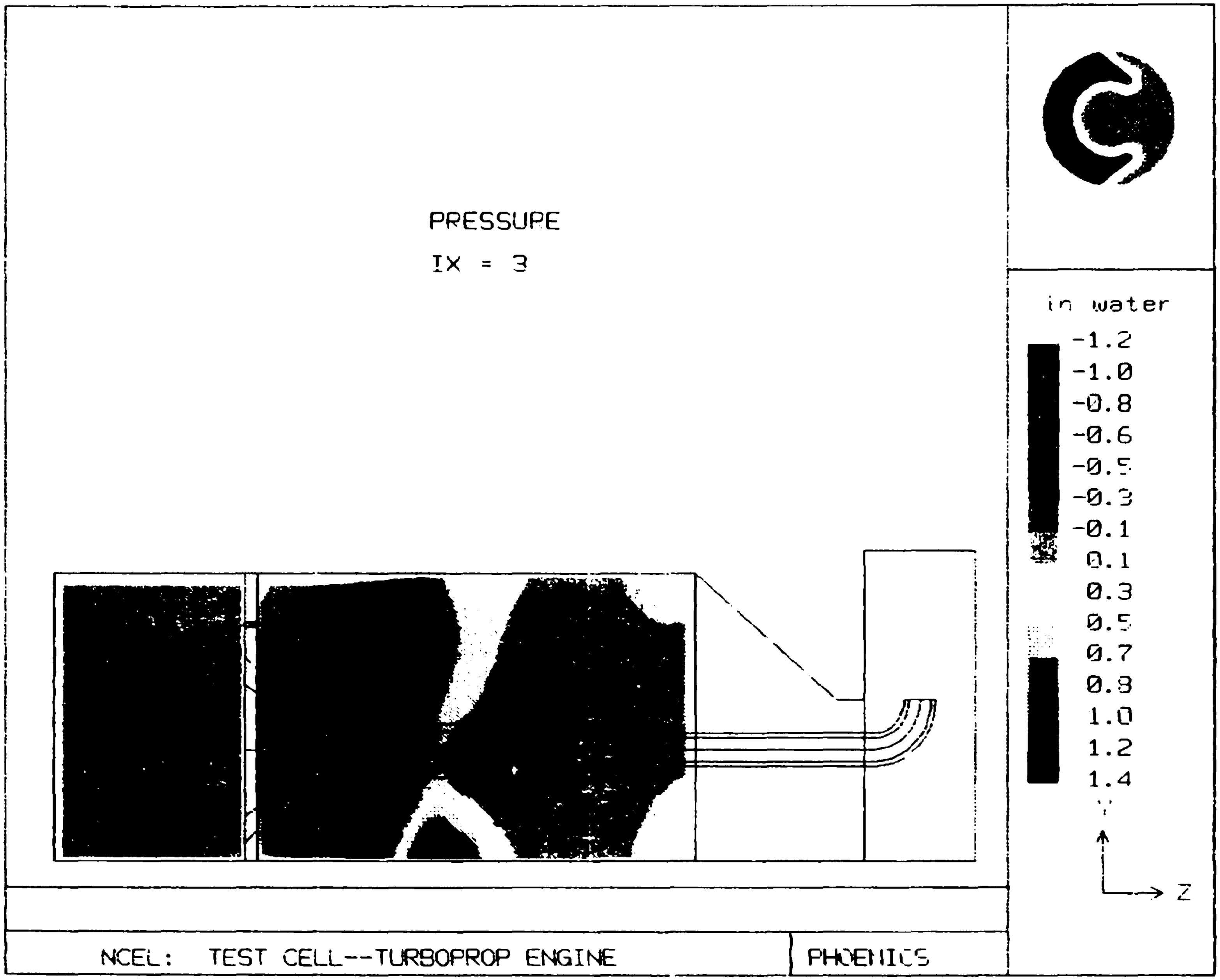


Figure 17 Pressure (Vertical Plane Near Wall)

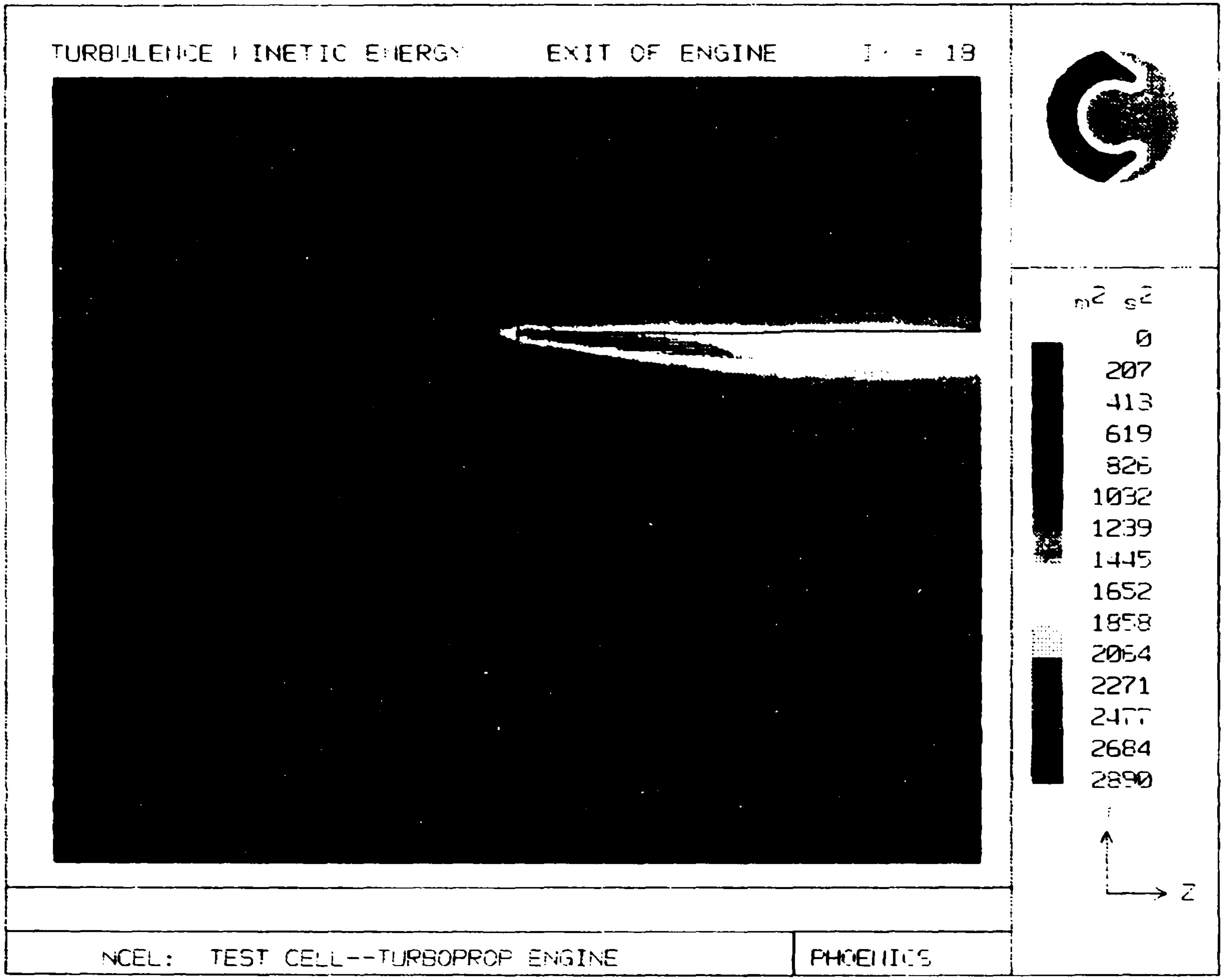


Figure 18 Turbulence Kinetic Energy Exit of Engine (Vertical Plane)

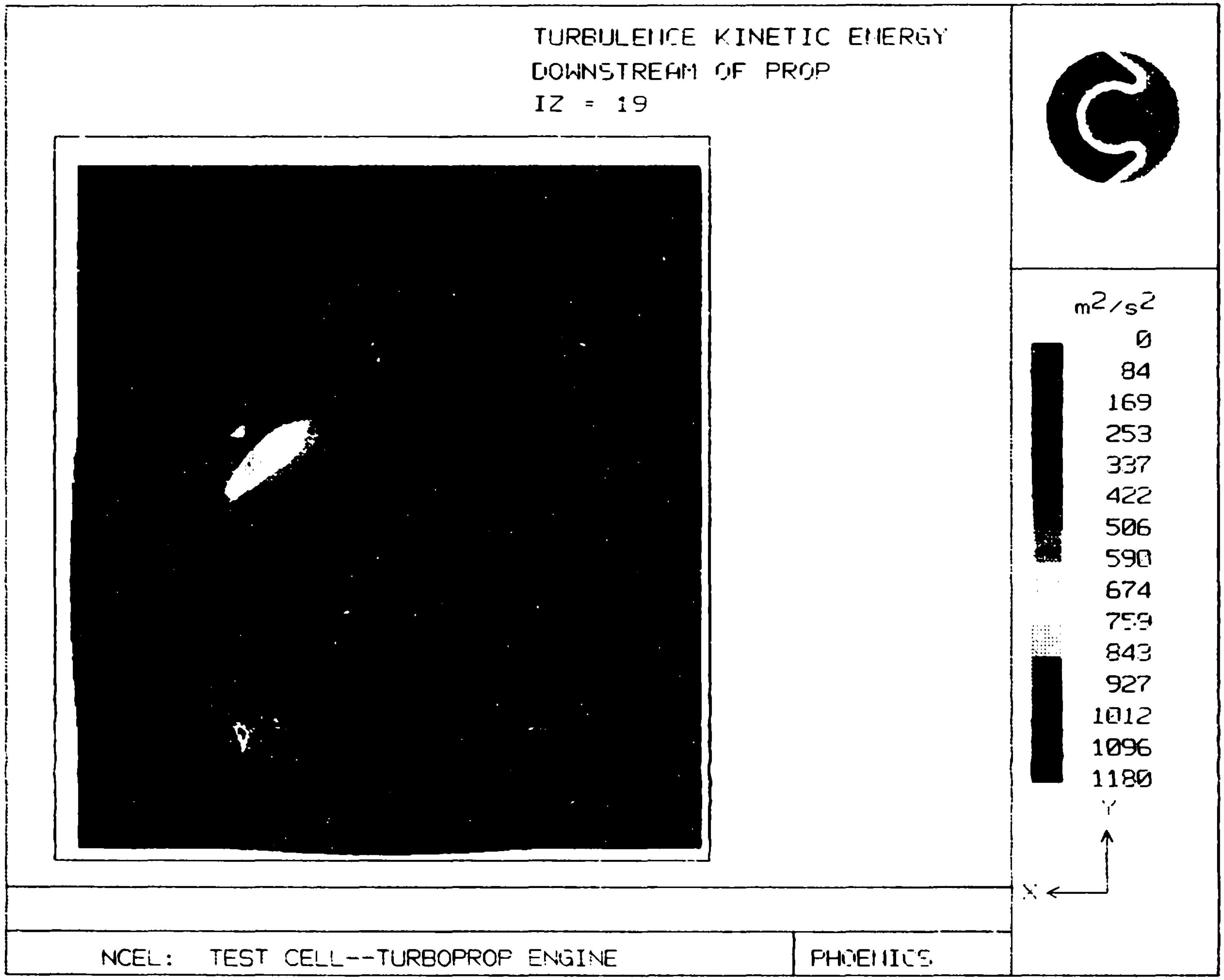
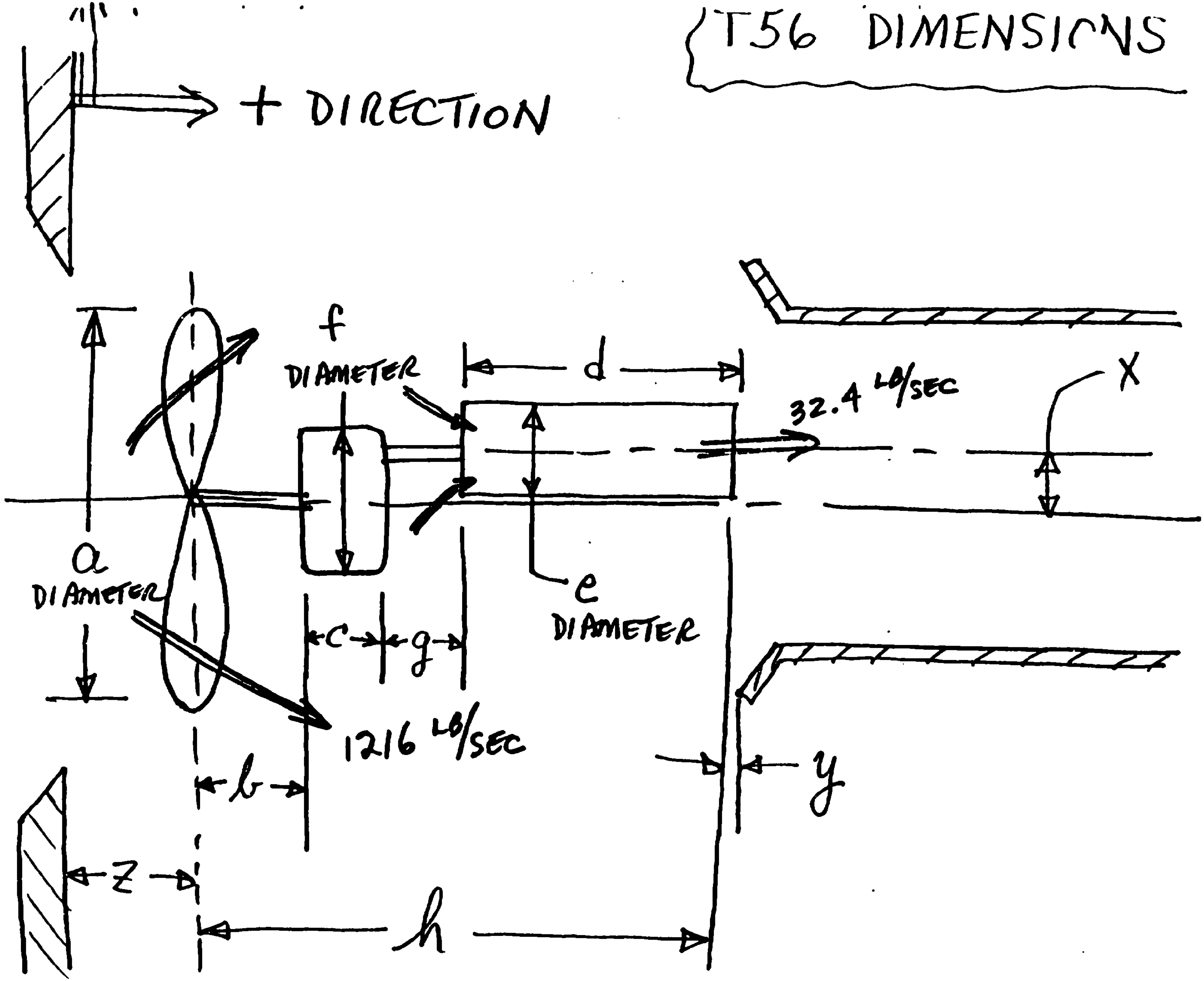


Figure 19 Turbulence Kinetic Energy Down Stream of Prop (Cross Plane)

APPENDIX A

T56 DIMENSIONS



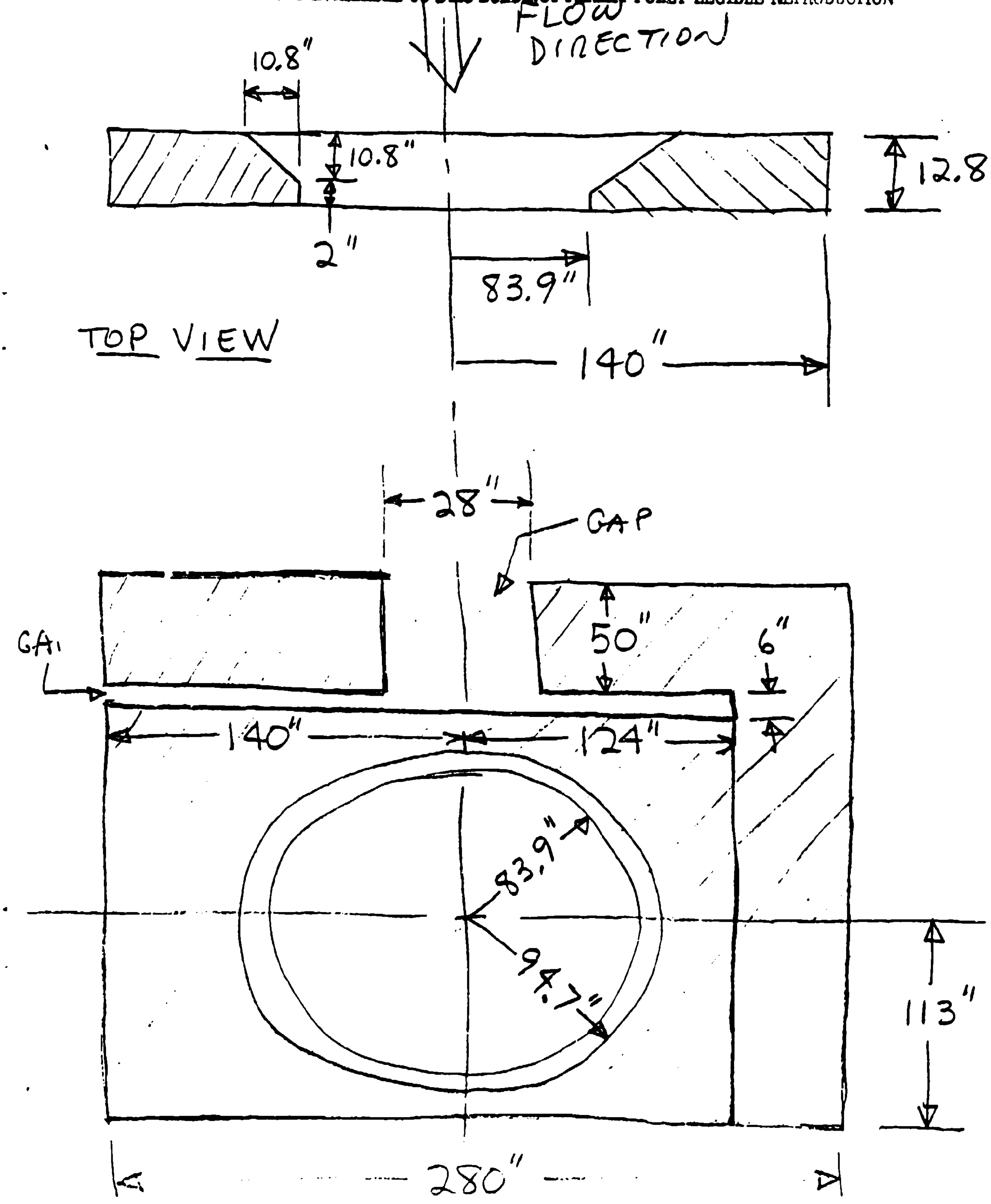
T56 DIMENSIONS

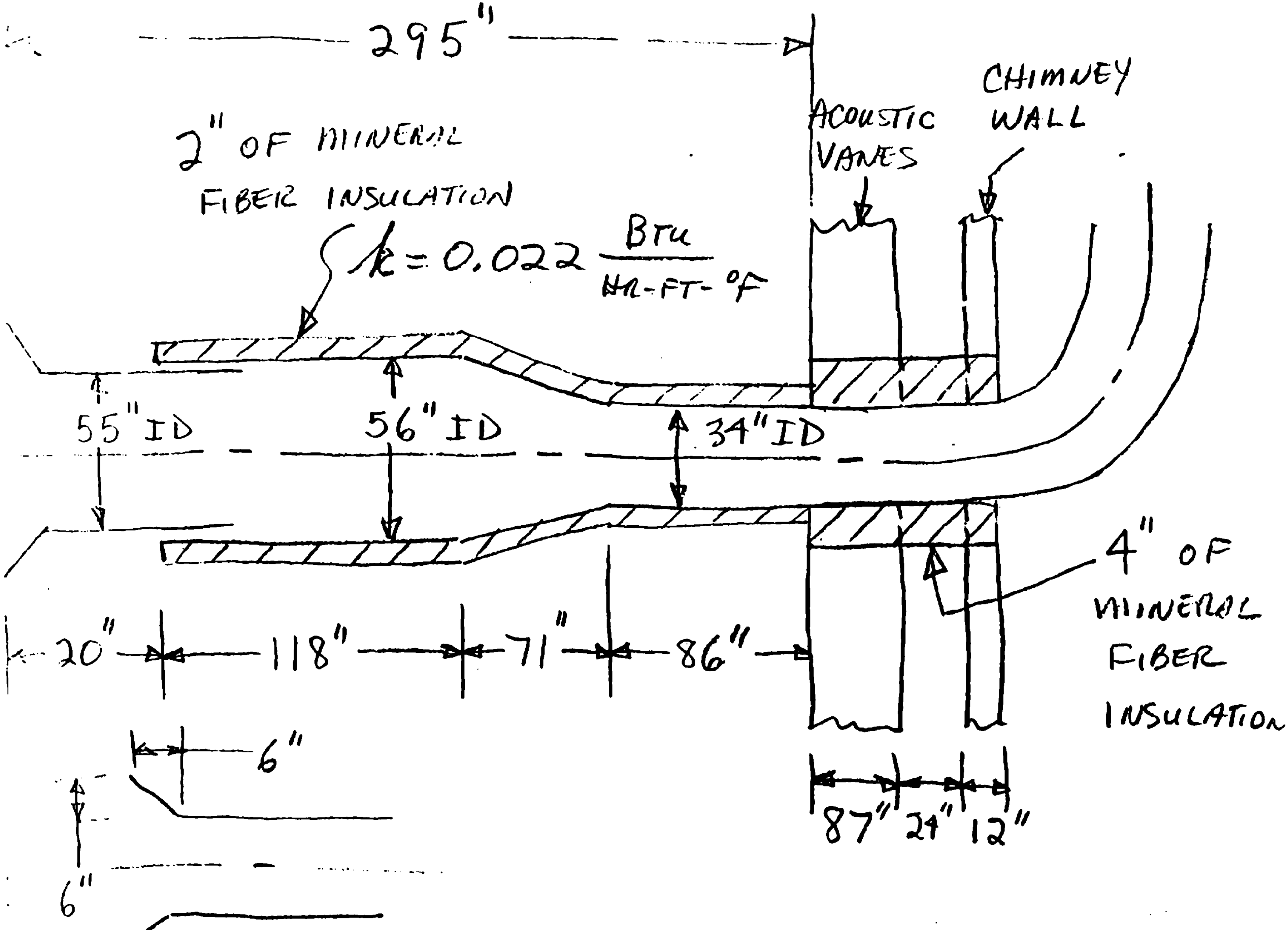
NOTATION	BASE (INCHES)	VARIATION
a PROP DIA	156	HIGHLY VARIABLE
b	14	*
c	20	*
d	96	VARIABLE
e NOZZLE DIA	18	VARIABLE
f REDUCTION GEAR DIA	27 ***	*
g	31	MAY BE IMPORTANT?
h	161	$h \le 177'' + 6''$
X	+9.	$-12'' \le X \le +12''$
Y	0.	$-16'' \le Y \le +6''$ **
Z	16.	$0. \le Z \le 22''$

* PROBABLY WILL HAVE LITTLE EFFECT ON FLOWS?

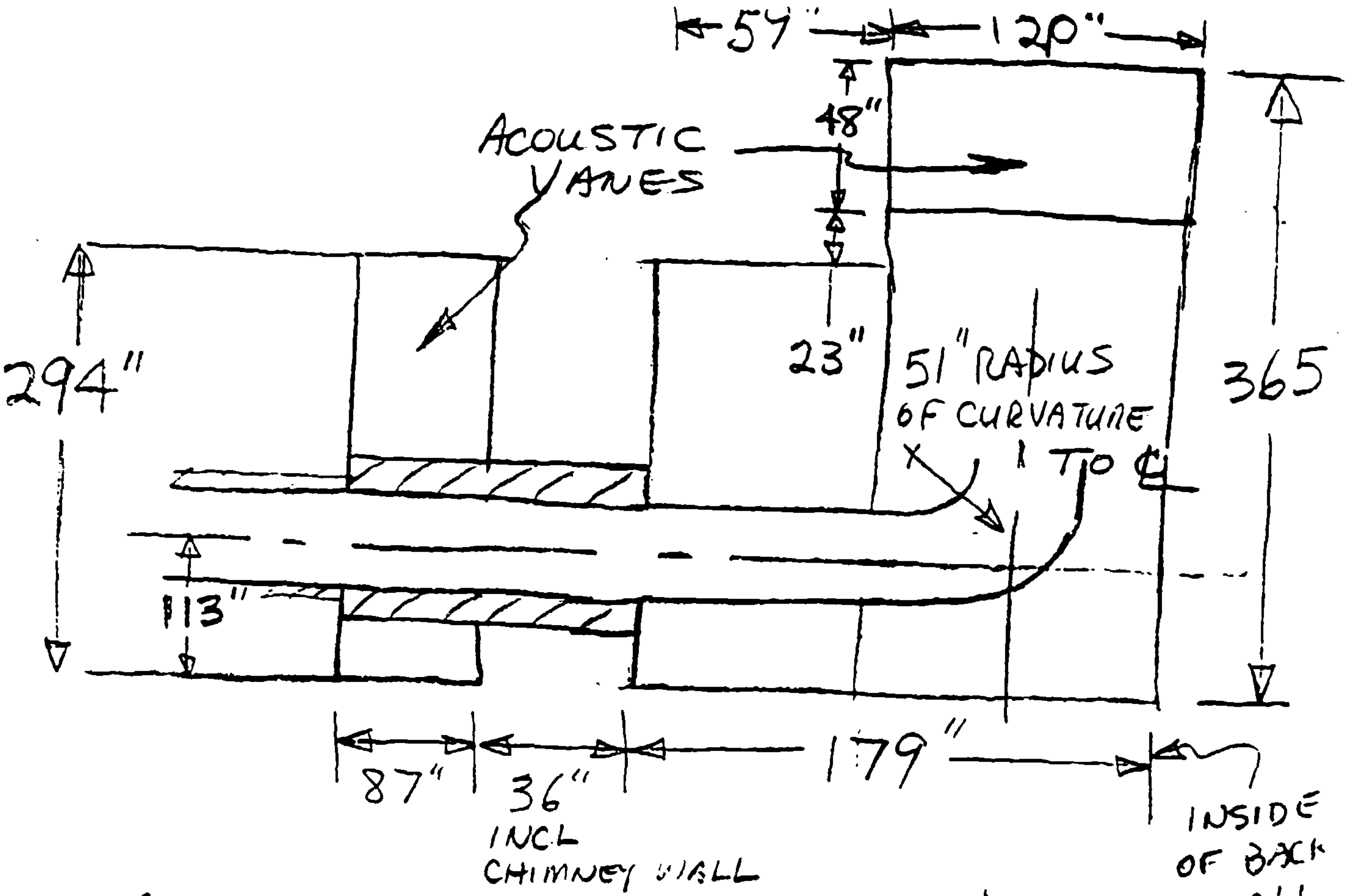
** NOT MORE THAN 6" PENETRATION OF AUG TUBE

*** ASSUME REDUCTION GEAR ALWAYS SYMMETRIC WITH RESPECT TO PROPELLER AXIS OF ROTATION?

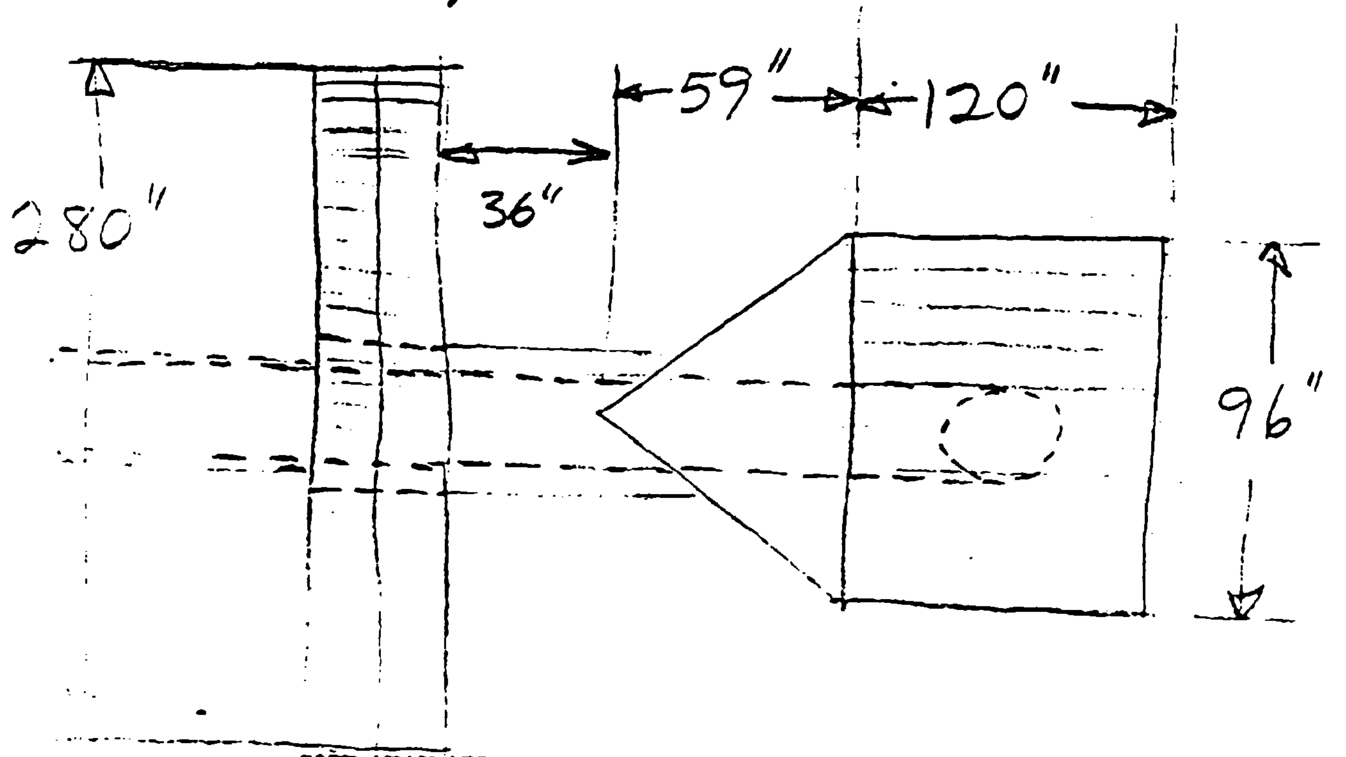

FLOW
DIRECTION
10.8"
10.8"
12.8
2"
83.9"
TOP VIEW
140"
28"
GAP
GA
50"
6"
140"
124"
83.9"
94.7"
113"
280"



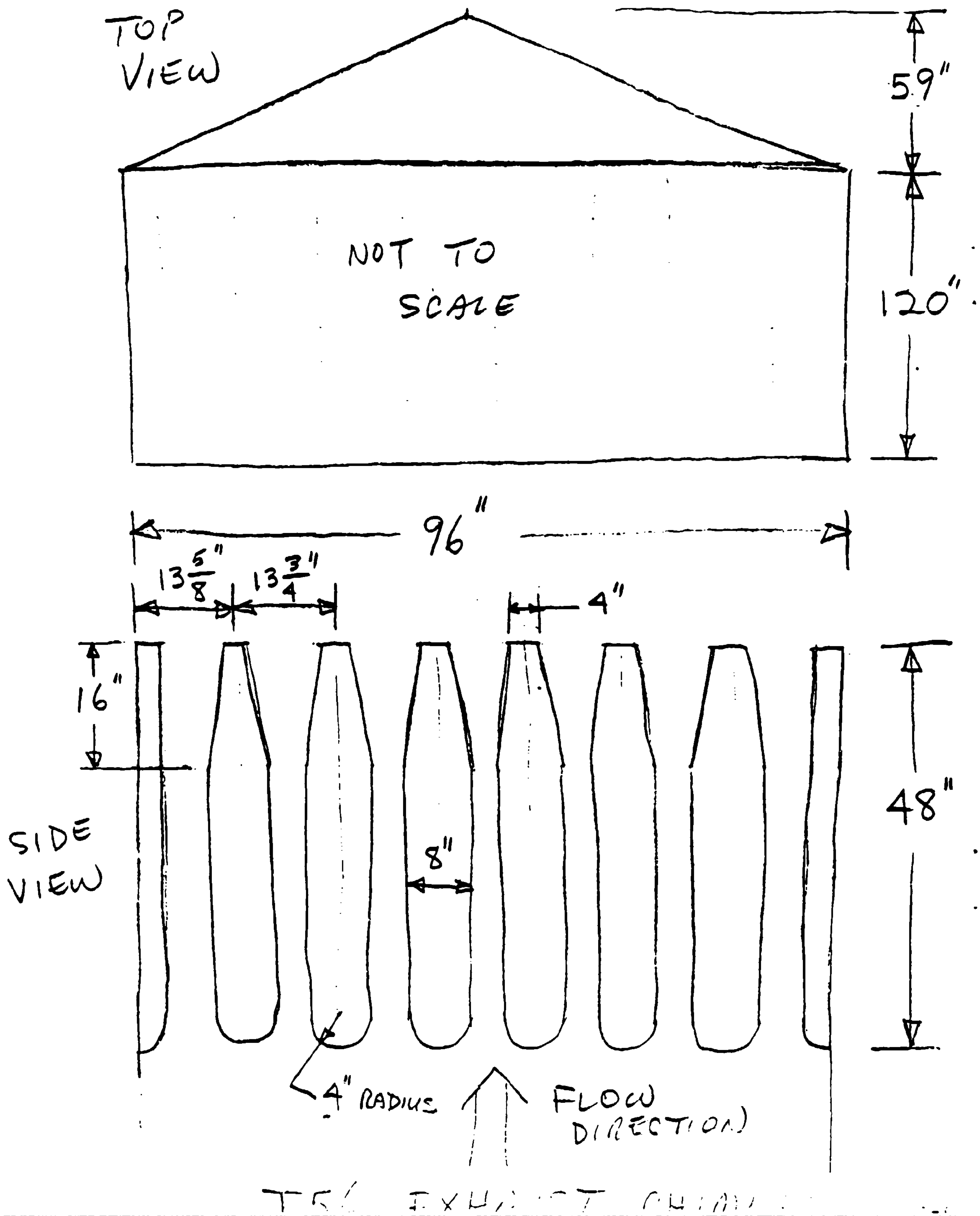
AUGMENTER TUBE



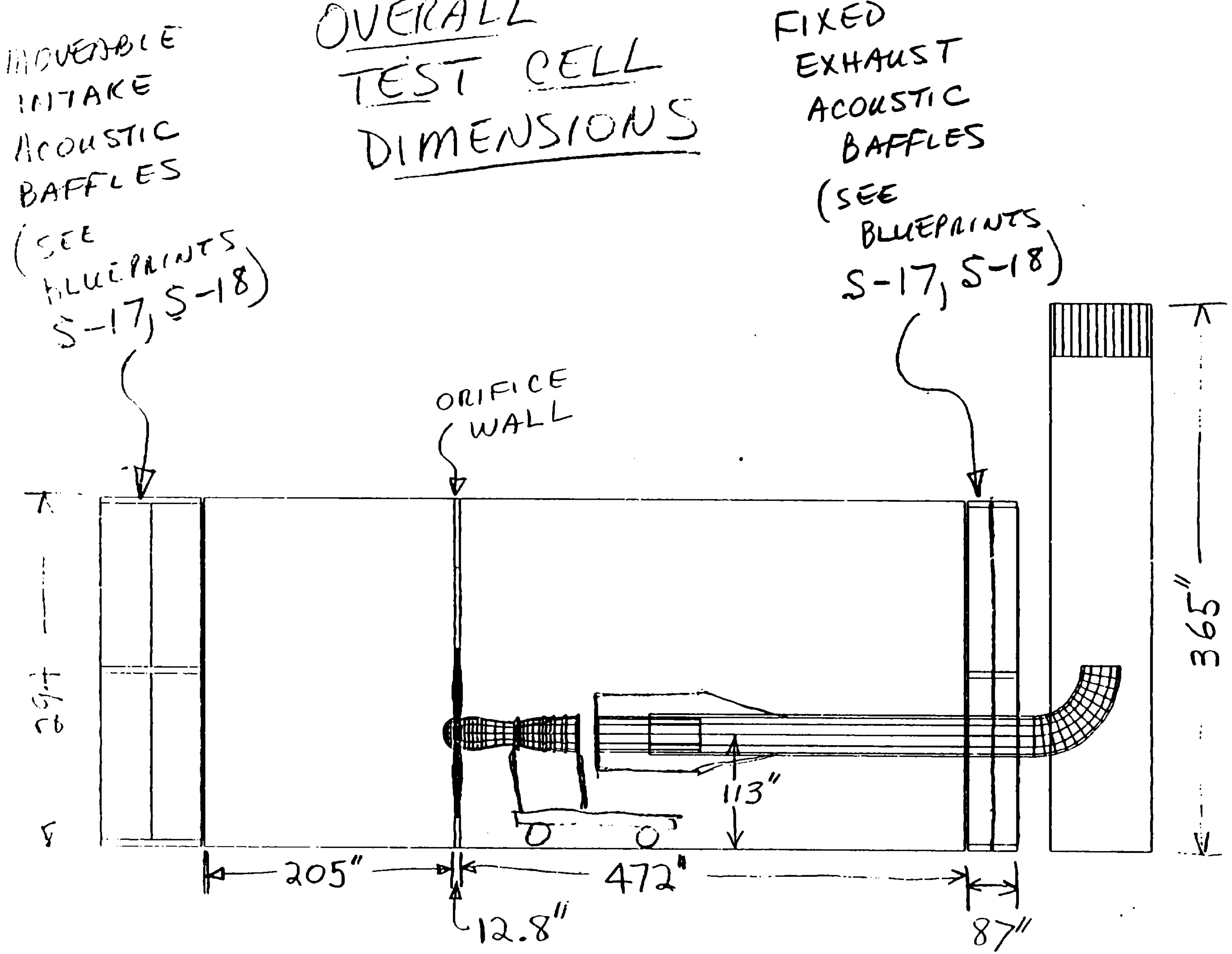
CHIMNEY, SIDE VIEW



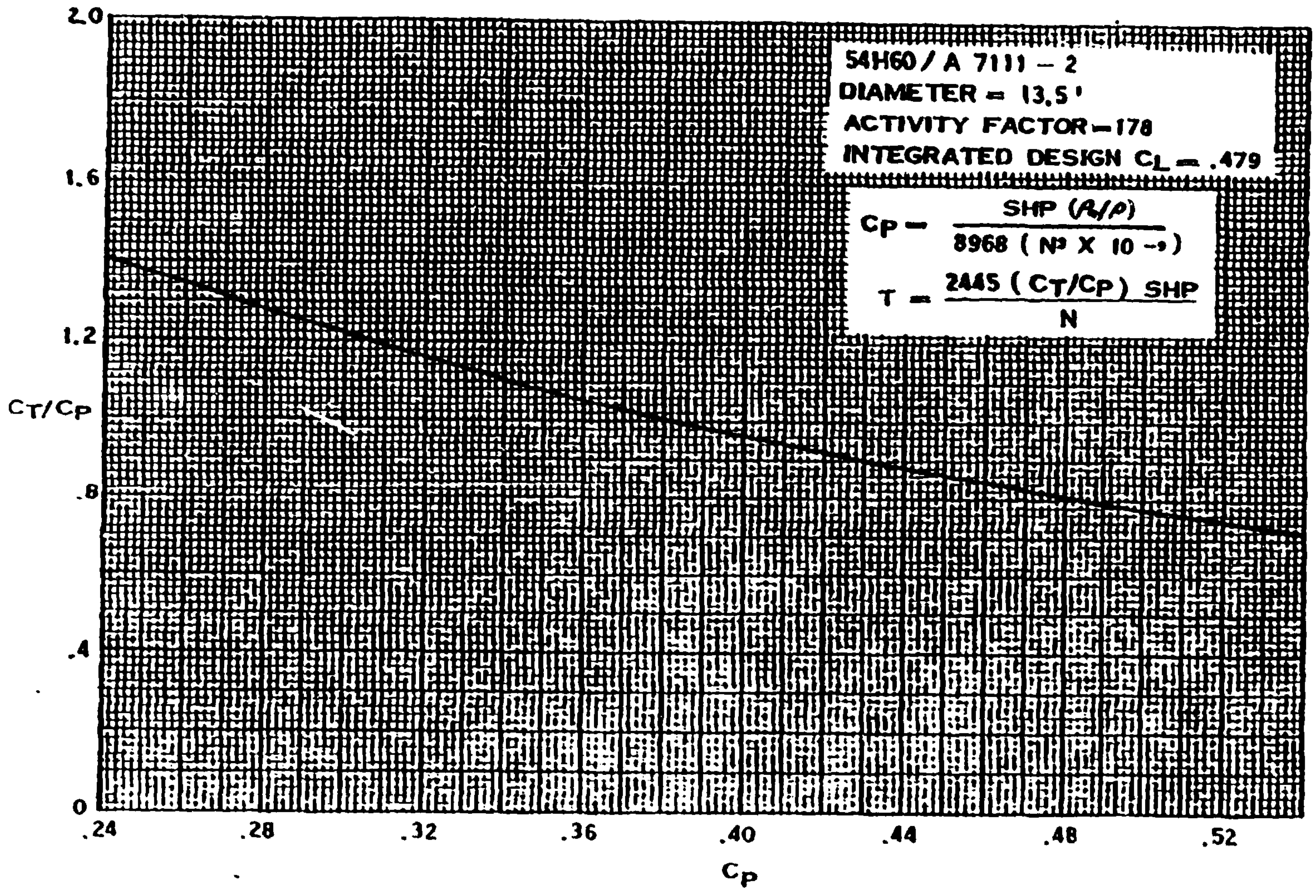
COPY AVAILABLE TO DTIC DOES NOT PERMIT FULLY LEGIBLE REPRODUCTION



OVERALL TEST CELL DIMENSIONS



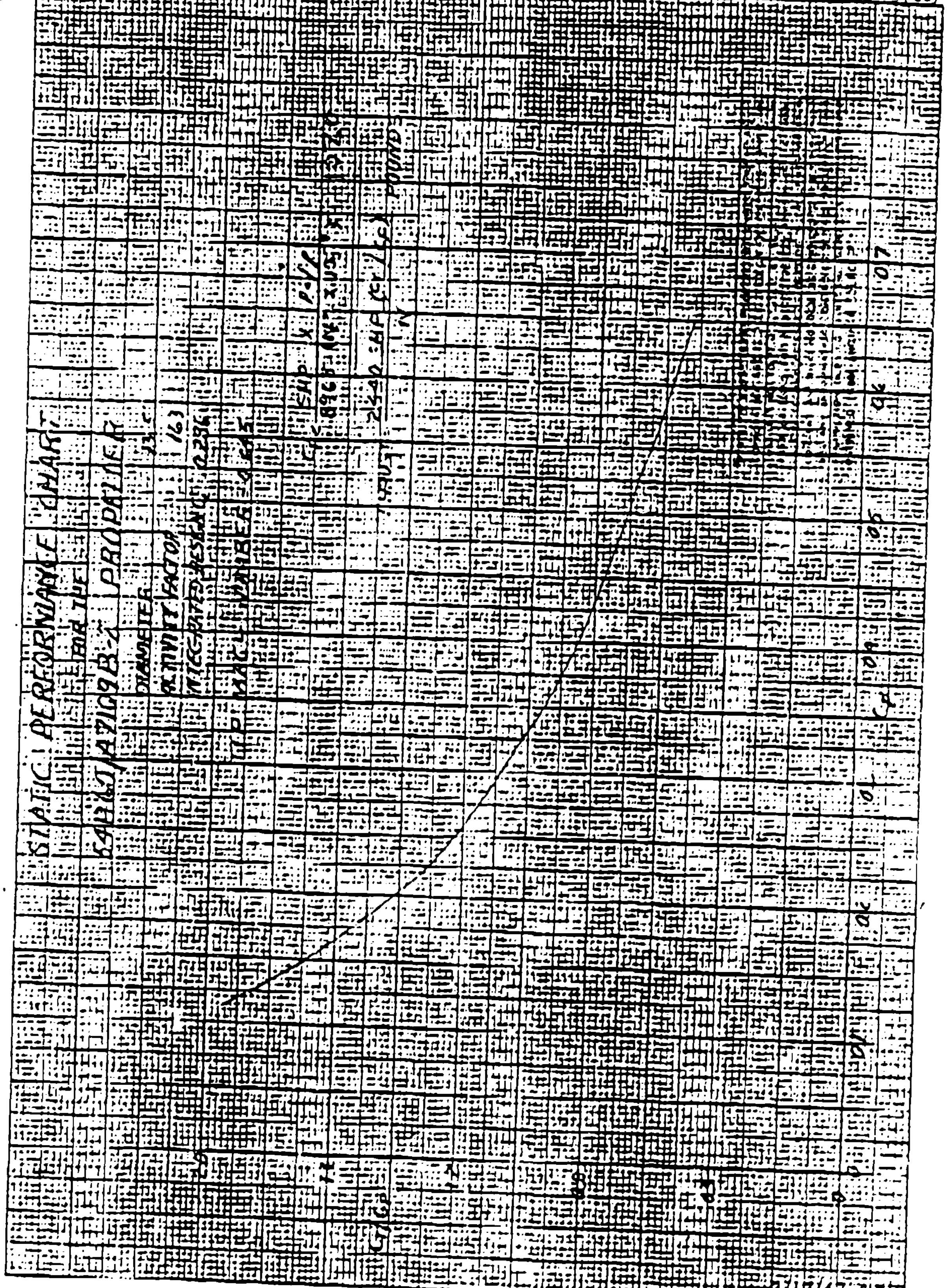
APPENDIX B



STATIC PERFORMANCE CHART

FIGURE 1

CALC. NO. 2311C(20)



HAMILTON STANDARD CURVE NO. P-30362

Hamilton Standard/United Technologies
Facsimile Transmission

From: **Robert W. Menthe**
Chief – Aerodynamics
Phone: (203) 654-5306
Fax: (203) 654-2659

To: Bill Mahaffey

Fax: 205 830 2628 Number of pages: 14

Message:

attached are approximate equations for computation of the average flow behind a static propeller. Make sure the dimension (units) of the density and power are such that the velocity has the correct dimension (units). Hope this is of some help.

Also for ref. $C_P = P/\rho n^3 D^5$

$C_T = T/\rho n^2 D^4$

Bob

10/14/92

Approximate Equations for Calculation of Average Flow behind a Static (V=0) Propeller.

a_p = axial induced velocity at the propeller disk

$$= \sqrt{\frac{T}{2\rho A_p}}$$

T = Thrust, ρ = density

$A_p = \pi D^2/4$, D = Prop Diameter

$$a_z/a_p = 1 + \frac{2z/D}{\sqrt{1+(2z/D)^2}}$$

, z = distance from Prop
+ downstream
- upstream

u_p = circumferential induced velocity at the prop. disk

$$= \frac{P}{\rho A_p a_p \pi n \bar{D}}$$

P = Power

n = rev/unit time

$\bar{D} = 0.75 \cdot D$

$u_z = 0 \quad -\infty < z < 0$ (upstream)

$u_z/u_p = \sqrt{a_z/a_p} \quad 0 \le z < \infty$ (downstream)

DISTRIBUTION QUESTIONNAIRE

The Naval Civil Engineering Laboratory is revising its primary distribution lists.

SUBJECT CATEGORIES

1 **SHORE FACILITIES**
1A Construction methods and materials (including corrosion control, coatings)
1B Waterfront structures (maintenance/deterioration control)
1C Utilities (including power conditioning)
1D Explosives safety
1E Aviation Engineering Test Facilities
1F Fire prevention and control
1G Antenna technology
1H Structural analysis and design (including numerical and computer techniques)
1J Protective construction (including hardened shelters, shock and vibration studies)
1K Soil/rock mechanics
1L Airfields and pavements
1M Physical security
2 **ADVANCED BASE AND AMPHIBIOUS FACILITIES**
2A Base facilities (including shelters, power generation, water supplies)
2B Expedient roads/airfields/bridges
2C Over-the-beach operations (including breakwaters, wave forces)
2D POL storage, transfer, and distribution
2E Polar engineering
3 **ENERGY/POWER GENERATION**
3A Thermal conservation (thermal engineering of buildings, HVAC systems, energy loss measurement, power generation)
3B Controls and electrical conservation (electrical systems, energy monitoring and control systems)
3C Fuel flexibility (liquid fuels, coal utilization, energy from solid waste)
3D Alternate energy source (geothermal power, photovoltaic power systems, solar systems, wind systems, energy storage systems)
3E Site data and systems integration (energy resource data, integrating energy systems)
3F EMCS design
4 **ENVIRONMENTAL PROTECTION**
4A Solid waste management
4B Hazardous/toxic materials management
4C Waterwaste management and sanitary engineering
4D Oil pollution removal and recovery
4E Air pollution
4F Noise abatement
5 **OCEAN ENGINEERING**
5A Seafloor soils and foundations
5B Seafloor construction systems and operations (including diver and manipulator tools)
5C Undersea structures and materials
5D Anchors and moorings
5E Undersea power systems, electromechanical cables, and connectors
5F Pressure vessel facilities
5G Physical environment (including site surveying)
5H Ocean-based concrete structures
5J Hyperbaric chambers
5K Undersea cable dynamics
ARMY FEAP
BDG Shore Facilities
NRG Energy
ENV Environmental/Natural Responses
MGT Management
PRR Pavements/Railroads

TYPES OF DOCUMENTS

D – Techdata Sheets; R – Technical Reports and Technical Notes; G – NCEL Guides and Abstracts; I – Index to TDS; U – User Guides; ☐ None - remove my name

Old Address:

____

____

____

____

Telephone No.: ________________

New Address:

____

____

____

____

Telephone No.: ________________

INSTRUCTIONS

The Naval Civil Engineering Laboratory has revised its primary distribution lists. To help us verify our records and update our data base, please do the following:

- Add - circle number on list
- Remove my name from all your lists - check box on list.
- Change my address - line out incorrect line and write in correction (DO NOT REMOVE LABEL).
- Number of copies should be entered after the title of the subject categories you select.
- Are we sending you the correct type of document? If not, circle the type(s) of document(s) you want to receive listed on the back of this card.

Fold on line, staple, and drop in mail.

DEPARTMENT OF THE NAVY
Naval Civil Engineering Laboratory
560 Laboratory Drive
Port Hueneme CA 93043-4328

Official Business
Penalty for Private Use, $300

NO POSTAGE NECESSARY IF MAILED IN THE UNITED STATES

BUSINESS REPLY CARD
FIRST CLASS PERMIT NO. 12503 WASH D.C.

POSTAGE WILL BE PAID BY ADDRESSEE

COMMANDING OFFICER
CODE L34
560 LABORATORY DRIVE
NAVAL CIVIL ENGINEERING LABORATORY
PORT HUENEME CA 93043-4328

NCEL DOCUMENT EVALUATION

You are number one with us; how do we rate with you?

We at NCEL want to provide you our customer the best possible reports but we need your help. Therefore, I ask you to please take the time from your busy schedule to fill out this questionnaire. Your response will assist us in providing the best reports possible for our users. I wish to thank you in advance for your assistance. I assure you that the information you provide will help us to be more responsive to your future needs.

R. N. Storer

R. N. STORER, Ph.D, P.E.
Technical Director

DOCUMENT NO. __________ TITLE OF DOCUMENT: ______________________________

____

Date: ____________ Respondent Organization : ______________________________

Name: ______________________________ Activity Code: ____________________
Phone: ______________________________ Grade/Rank: ____________________

Category (*please check*):

Sponsor _____ User _____ Proponent _____ Other (*Specify*) ____________________

Please answer on your behalf only; not on your organization's. Please check (*use an X*) only the block that most closely describes your attitude or feeling toward that statement:

SA Strongly Agree A Agree O Neutral D Disagree SD Strongly Disagree

---

	SA	A	N	D	SD
1. The technical quality of the report is comparable to most of my other sources of technical information.	( )	( )	( )	( )	( )
2. The report will make significant improvements in the cost and or performance of my operation.	( )	( )	( )	( )	( )
3. The report acknowledges related work accomplished by others.	( )	( )	( )	( )	( )
4. The report is well formatted.	( )	( )	( )	( )	( )
5. The report is clearly written.	( )	( )	( )	( )	( )
6. The conclusions and recommendations are clear and directly supported by the contents of the report.	( )	( )	( )	( )	( )
7. The graphics, tables, and photographs are well done.	( )	( )	( )	( )	( )

Do you wish to continue getting NCEL reports? ☐ YES ☐ NO

Please add any comments (e.g., in what ways can we improve the quality of our reports?) on the back of this form.

Comments:

Fold on line, staple, and drop in mail.

DEPARTMENT OF THE NAVY
Naval Civil Engineering Laboratory
560 Laboratory Drive
Port Hueneme CA 93043-4328

Official Business
Penalty for Private Use, $300

NO POSTAGE NECESSARY IF MAILED IN THE UNITED STATES

BUSINESS REPLY CARD

FIRST CLASS PERMIT NO. 12503 WASH D.C.

POSTAGE WILL BE PAID BY ADDRESSEE

COMMANDING OFFICER
CODE L03
560 LABORATORY DRIVE
NAVAL CIVIL ENGINEERING LABORATORY
PORT HUENEME CA 93043-4328

END

FILMED

DATE: 8-93

DTIC